# Solutions Manual for
# Fluid Mechanics: Fundamentals and Applications
# by Çengel & Cimbala

---

# CHAPTER 1
# INTRODUCTION AND BASIC CONCEPTS

---

## Introduction, Classification, and System

### 1-1C

**Solution** We are to define internal, external, and open-channel flows.

***Analysis*** *External flow* is the **flow of an unbounded fluid over a surface** such as a plate, a wire, or a pipe. The flow in a pipe or duct is *internal flow* if the **fluid is completely bounded by solid surfaces**. The flow of liquids in a pipe is called *open-channel flow* if the pipe is **partially filled with the liquid and there is a free surface**, such as the flow of water in rivers and irrigation ditches.

***Discussion*** As we shall see in later chapters, there different approximations are used in the analysis of fluid flows based on their classification.

### 1-2C

**Solution** We are to define incompressible and compressible flow, and discuss fluid compressibility.

***Analysis*** A fluid flow during which the **density of the fluid remains nearly constant** is called *incompressible flow*. A flow in which **density varies significantly** is called *compressible flow*. A fluid whose density is practically independent of pressure (such as a liquid) is commonly referred to as an "incompressible fluid," although it is more proper to refer to incompressible *flow*. The flow of compressible fluid (such as air) does not necessarily need to be treated as compressible since the density of a compressible fluid may still remain nearly constant during flow – especially flow at low speeds.

***Discussion*** It turns out that the Mach number is the critical parameter to determine whether the flow of a gas can be approximated as an incompressible flow. If Ma is less than about 0.3, the incompressible approximation yields results that are in error by less than a couple percent.

### 1-3C

**Solution** We are to define the no-slip condition and its cause.

***Analysis*** A **fluid in direct contact with a solid surface sticks to the surface and there is no slip**. This is known as the *no-slip condition*, and it is due to the *viscosity* of the fluid.

***Discussion*** There is no such thing as an inviscid fluid, since all fluids have viscosity.

### 1-4C

**Solution** We are to define forced flow and discuss the difference between forced and natural flow. We are also to discuss whether wind-driven flows are forced or natural.

***Analysis*** In *forced flow*, the fluid is forced to flow over a surface or in a tube **by *external* means** such as a pump or a fan. In *natural flow*, any fluid motion is caused by natural means such as the buoyancy effect that manifests itself as the rise of the warmer fluid and the fall of the cooler fluid. **The flow caused by winds is natural flow for the earth, but it is forced flow for bodies subjected to the winds** since for the body it makes no difference whether the air motion is caused by a fan or by the winds.

***Discussion*** As seen here, the classification of forced vs. natural flow may depend on your frame of reference.

**1-5C**

**Solution** We are to define a boundary layer, and discuss its cause.

***Analysis*** When a fluid stream encounters a solid surface that is at rest, the fluid velocity assumes a value of zero at that surface. The velocity then varies from zero at the surface to the freestream value sufficiently far from the surface. The **region of flow in which the velocity gradients are significant and frictional effects are important** is called the *boundary layer*. The development of a boundary layer is caused by the *no-slip condition*.

***Discussion*** As we shall see later, flow within a boundary layer is *rotational* (individual fluid particles rotate), while that outside the boundary layer is typically *irrotational* (individual fluid particles move, but do not rotate).

---

**1-6C**

**Solution** We are to discuss the differences between classical and statistical approaches.

***Analysis*** The *classical approach* is **a *macroscopic* approach**, based on experiments or analysis of the gross behavior of a fluid, without knowledge of individual molecules, whereas the *statistical approach* is a ***microscopic* approach** based on the average behavior of large groups of individual molecules.

***Discussion*** The classical approach is easier and much more common in fluid flow analysis.

---

**1-7C**

**Solution** We are to define a steady-flow process.

***Analysis*** A process is said to be *steady* if it involves **no changes with time** anywhere within the system or at the system boundaries.

***Discussion*** The opposite of steady flow is *unsteady flow*, which involves changes with time.

---

**1-8C**

**Solution** We are to define stress, normal stress, shear stress, and pressure.

***Analysis*** *Stress* is defined as **force per unit area**, and is determined by dividing the force by the area upon which it acts. The **normal component of a force acting on a surface per unit area** is called the *normal stress*, and the **tangential component of a force acting on a surface per unit area** is called *shear stress*. In a fluid at rest, the normal stress is called *pressure*.

***Discussion*** Fluids in motion may have additional normal stresses, but when a fluid is at rest, the only normal stress is the pressure.

---

**1-9C**

**Solution** We are to define system, surroundings, and boundary.

***Analysis*** A *system* is defined as a **quantity of matter or a region in space chosen for study**. The mass or **region outside the system** is called the *surroundings***.** The real or imaginary **surface that separates the system from its surroundings** is called the *boundary*.

***Discussion*** Some authors like to define *closed systems* and *open systems*, while others use the notation "system" to mean a closed system and "control volume" to mean an open system. This has been a source of confusion for students for many years. [See the next question for further discussion about this.]

---

**1-10C**

**Solution** We are to discuss when a system is considered closed or open.

***Analysis*** Systems may be considered to be *closed* or *open*, depending on whether a fixed mass or a volume in space is chosen for study. A *closed system* (also known as a *control mass* or simply a *system*) consists of a **fixed amount of mass, and no mass can cross its boundary**. An *open system***,** or a *control volume*, is a **properly selected region in space**.

***Discussion*** In thermodynamics, it is more common to use the terms *open system* and *closed system*, but in fluid mechanics, it is more common to use the terms *system* and *control volume* to mean the same things, respectively.

## Mass, Force, and Units

**1-11C**

**Solution** We are to discuss the difference between pound-mass and pound-force.

***Analysis*** *Pound-mass* lbm is the **mass unit in English system** whereas *pound-force* lbf is the **force unit in the English system**. One pound-force is the force required to accelerate a mass of 32.174 lbm by 1 $ft/s^2$. In other words, the weight of a 1-lbm mass at sea level on earth is 1 lbf.

***Discussion*** It is *not* proper to say that one lbm is equal to one lbf since the two units have different dimensions.

**1-12C**

**Solution** We are to discuss the difference between kg-mass and kg-force.

***Analysis*** The unit *kilogram* (kg) is the **mass unit in the SI system**, and it is sometimes called *kg-mass*, whereas *kg-force* (kgf) is a **force unit**. One kg-force is the force required to accelerate a 1-kg mass by 9.807 $m/s^2$. In other words, the weight of 1-kg mass at sea level on earth is 1 kg-force.

***Discussion*** It is *not* proper to say that one kg-mass is equal to one kg-force since the two units have different dimensions.

**1-13C**

**Solution** We are to calculate the net force on a car cruising at constant velocity.

***Analysis*** There is no acceleration, thus **the net force is zero in both cases**.

***Discussion*** By Newton's second law, the force on an object is directly proportional to its acceleration. If there is zero acceleration, there must be zero net force.

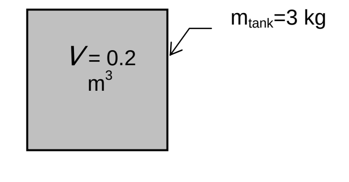
**1-14**

**Solution** A plastic tank is filled with water. The weight of the combined system is to be determined.

***Assumptions*** The density of water is constant throughout.

***Properties*** The density of water is given to be $\rho$ = 1000 kg/m$^3$.

***Analysis*** The mass of the water in the tank and the total mass are




$$m_w = \rho V = (1000 \text{ kg/m}^3)(0.2 \text{ m}^3) = 200 \text{ kg}$$
$$m_{\text{total}} = m_w + m_{tank} = 200 + 3 = 203 \text{ kg}$$

Thus,

$$W = mg = (203 \text{ kg})(9.81 \text{ m/s}^2)\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = 1991 \text{ N} \cong \mathbf{1990 \text{ N}}$$

where we give the final answer to three significant digits.

***Discussion*** Note the unity conversion factor in the above equation.

---

**1-15**

**Solution** The interior dimensions of a room are given. The mass and weight of the air in the room are to be determined.

***Assumptions*** The density of air is constant throughout the room.

***Properties*** The density of air is given to be $\rho$ = 1.16 kg/m$^3$.

***Analysis*** The mass of the air in the room is

ROOM
AIR
6X6X8 m$^3$

$$m = \rho V = (1.16 \text{ kg/m}^3)(6\times6\times8 \text{ m}^3) = 334.1 \text{ kg} \cong \mathbf{334 \text{ kg}}$$

Thus,

$$W = mg = (334.1 \text{ kg})(9.81 \text{ m/s}^2)\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = 3277 \text{ N} \cong \mathbf{3280 \text{ N}}$$

***Discussion*** Note that we round our final answers to three significant digits, but use extra digit(s) in intermediate calculations. Considering that the mass of an average man is about 70 to 90 kg, the mass of air in the room is probably larger than you might have expected.

---

**1-16**

**Solution** The variation of gravitational acceleration above sea level is given as a function of altitude. The height at which the weight of a body decreases by 1% is to be determined.

***Analysis*** The weight of a body at the elevation *z* can be expressed as

$$W = mg = m(9.807 - 3.32\times10^{-6}z)$$

In our case,

$$W = 0.99W_s = 0.99mg_s = 0.99(m)(9.807)$$

Substituting,

$$0.99(9.807) = (9.807 - 3.32\times10^{-6}z) \longrightarrow z = 29{,}540 \text{ m} \cong \mathbf{29{,}500 \text{ m}}$$

*z*
0
Sea level

where we have rounded off the final answer to three significant digits.

***Discussion*** This is more than three times higher than the altitude at which a typical commercial jet flies, which is about 30,000 ft (9140 m). So, flying in a jet is not a good way to lose weight – diet and exercise are always the best bet.

---

**1-17E**

**Solution** An astronaut takes his scales with him to the moon. It is to be determined how much he weighs on the spring and beam scales on the moon.

***Analysis***

(*a*) A spring scale measures weight, which is the local gravitational force applied on a body:

$$W = mg = (150 \text{ lbm})(5.48 \text{ ft/s}^2)\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm} \cdot \text{ft/s}^2}\right) = \mathbf{25.5 \text{ lbf}}$$

(*b*) A beam scale compares masses and thus is not affected by the variations in gravitational acceleration. The beam scale reads what it reads on earth,

$$W = \mathbf{150 \text{ lbf}}$$

***Discussion*** The beam scale may be marked in units of weight (lbf), but it really compares mass, not weight. Which scale would you consider to be more accurate?

---

**1-18**

**Solution** The acceleration of an aircraft is given in *g*'s. The net upward force acting on a man in the aircraft is to be determined.

***Analysis*** From Newton's second law, the applied force is

$$F = ma = m(6 \text{ g}) = (90 \text{ kg})(6 \times 9.81 \text{ m/s}^2)\left(\frac{1 \text{ N}}{1 \text{ kg} \cdot \text{m/s}^2}\right) = 5297 \text{ N} \cong \mathbf{5300 \text{ N}}$$

where we have rounded off the final answer to three significant digits.

***Discussion*** The man feels like he is six times heavier than normal. You get a similar feeling when riding an elevator to the top of a tall building, although to a much lesser extent.

---

**1-19 [*Also solved by EES on enclosed CD*]**

**Solution** A rock is thrown upward with a specified force. The acceleration of the rock is to be determined.

***Analysis*** The weight of the rock is

$$W = mg = (5 \text{ kg})(9.79 \text{ m/s}^2)\left(\frac{1 \text{ N}}{1 \text{ kg} \cdot \text{m/s}^2}\right) = 48.95 \text{ N} \cong \mathbf{49.0 \text{ N}}$$

Then the net force that acts on the rock is

$$F_{net} = F_{up} - F_{down} = 150 - 48.95 = 101.05 \text{ N}$$

From Newton's second law, the acceleration of the rock becomes

$$a = \frac{F}{m} = \frac{101.05 \text{ N}}{5 \text{ kg}}\left(\frac{1 \text{ kg} \cdot \text{m/s}^2}{1 \text{ N}}\right) = \mathbf{20.2 \text{ m/s}^2}$$

Rock

***Discussion*** This acceleration is more than twice the acceleration at which it would fall (due to gravity) if dropped.

---


**1-20** 

**Solution** The previous problem is recalculated using EES. The entire EES solution is to be printed out, including the numerical results with proper units.

***Analysis*** The EES *Equations* window is printed below, followed by the *Solution* window.

```
W=m*g"[N]"
m=5"[kg]"
g=9.79"[m/s^2]"
"The force balance on the rock yields the net force acting on the rock as"
F_net = F_up - F_down"[N]"
F_up=150"[N]"
F_down=W"[N]"
"The acceleration of the rock is determined from Newton's second law."
F_net=a*m
"To Run the program, press F2 or click on the calculator icon from the Calculate menu"
```

**SOLUTION**
Variables in Main
a=20.21 [m/s^2]
F_down=48.95 [N]
F_net=101.1 [N]
F_up=150 [N]
g=9.79 [m/s^2]
m=5 [kg]
W=48.95 [N]

The final results are ***W* = 49.0 N** and ***a* = 20.2 m/s²**, to three significant digits, which agree with the results of the previous problem.

***Discussion*** Items in quotation marks in the EES Equation window are comments. Units are in square brackets.

**1-21**
**Solution** Gravitational acceleration *g* and thus the weight of bodies decreases with increasing elevation. The percent reduction in the weight of an airplane cruising at 13,000 m is to be determined.

***Properties*** The gravitational acceleration *g* is 9.807 $m/s^2$ at sea level and 9.767 $m/s^2$ at an altitude of 13,000 m.

***Analysis*** Weight is proportional to the gravitational acceleration *g*, and thus the percent reduction in weight is equivalent to the percent reduction in the gravitational acceleration, which is determined from

$$\%\text{ Reduction in weight} = \%\text{ Reduction in } g = \frac{\Delta g}{g}\times 100 = \frac{9.807-9.767}{9.807}\times 100 = \mathbf{0.41\%}$$

Therefore, **the airplane and the people in it will weigh 0.41% less at 13,000 m altitude**.

***Discussion*** Note that the weight loss at cruising altitudes is negligible. Sorry, but flying in an airplane is not a good way to lose weight. The best way to lose weight is to carefully control your diet, and to exercise.

## Modeling and Solving Problems, and Precision

### 1-22C

**Solution** We are to discuss the difference between accuracy and precision.

***Analysis*** *Accuracy* refers to the **closeness of the measured or calculated value to the true value** whereas *precision* represents the **number of significant digits or the closeness of different measurements of the same quantity to each other**. **A measurement or calculation can be very precise without being very accurate, and vice-versa**. When measuring the boiling temperature of pure water at standard atmospheric conditions, for example, a temperature measurement of 97.861°C is very precise, but not as accurate as the less precise measurement of 99.0°C.

***Discussion*** Accuracy and precision are often confused; both are important for quality engineering measurements.

### 1-23C

**Solution** We are to discuss the difference between analytical and experimental approaches.

***Analysis*** The *experimental approach* (**testing and taking measurements**) has the advantage of dealing with the actual physical system, and getting a physical value within the limits of experimental error. However, this approach is expensive, time consuming, and often impractical. The *analytical approach* (**analysis or calculations**) has the advantage that it is fast and inexpensive, but the results obtained are subject to the accuracy of the assumptions and idealizations made in the analysis.

***Discussion*** Most engineering designs require both analytical and experimental components, and both are important. Nowadays, computational fluid dynamics (CFD) is often used in place of pencil-and-paper analysis and/or experiments.

### 1-24C

**Solution** We are to discuss the importance of modeling in engineering.

***Analysis*** *Modeling* makes it possible to **predict the course of an event before it actually occurs, or to study various aspects of an event mathematically without actually running expensive and time-consuming experiments**. When preparing a mathematical model, all the variables that affect the phenomena are identified, reasonable assumptions and approximations are made, and the interdependence of these variables are studied. The relevant physical laws and principles are invoked, and the problem is formulated mathematically. Finally, the problem is solved using an appropriate approach, and the results are interpreted.

***Discussion*** In most cases of actual engineering design, the results are verified by experiment – usually by building a prototype. CFD is also being used more and more in the design process.

### 1-25C

**Solution** We are to discuss choosing a model.

***Analysis*** The right choice between a crude and complex model is usually **the *simplest* model that yields *adequate* results**. Preparing very accurate but complex models is not necessarily a better choice since such models are not much use to an analyst if they are very difficult and time consuming to solve. At a minimum, the model should reflect the essential features of the physical problem it represents.

***Discussion*** Cost is always an issue in engineering design, and "adequate" is often determined by cost.

**1-26C**

**Solution** We are to discuss how differential equations arise in the study of a physical problem.

***Analysis*** The description of most scientific problems involves equations that relate the *changes* in some key variables to each other, and the smaller the increment chosen in the changing variables, the more accurate the description. In **the limiting case of infinitesimal changes in variables**, we obtain *differential equations*, which provide precise mathematical formulations for the physical principles and laws by representing the rates of changes as *derivatives*.

***Discussion*** As we shall see in later chapters, the differential equations of fluid mechanics are known, but very difficult to solve except for very simple geometries. Computers are extremely helpful in this area.

---

**1-27C**

**Solution** We are to discuss the value of engineering software packages.

***Analysis*** *Software packages* **are of great value in engineering practice, and engineers today rely on software packages to solve large and complex problems quickly, and to perform optimization studies efficiently**. Despite the convenience and capability that engineering software packages offer, they are still just *tools*, and they cannot replace traditional engineering courses. They simply cause a shift in emphasis in the course material from mathematics to physics.

***Discussion*** While software packages save us time by reducing the amount of number-crunching, we must be careful to understand how they work and what they are doing, or else incorrect results can occur.

---

**1-28** 

**Solution** We are to determine a positive real root of the following equation using EES: $2x^3 - 10x^{0.5} - 3x = -3$.

***Analysis*** Using EES software, copy the following lines and paste on a blank EES screen to verify the solution:

```
2*x^3-10*x^0.5-3*x = -3
```

*Answer*: ***x* = 2.063** (using an initial guess of $x = 2$)

***Discussion*** To obtain the solution in EES, click on the icon that looks like a calculator, or Calculate-Solve.

---

**1-29** 

**Solution** We are to solve a system of 2 equations and 2 unknowns using EES.

***Analysis*** Using EES software, copy the following lines and paste on a blank EES screen to verify the solution:

```
x^3-y^2=7.75
3*x*y+y=3.5
```

*Answers*: ***x* = 2.0**, ***y* = 0.50**.

***Discussion*** To obtain the solution in EES, click on the icon that looks like a calculator, or Calculate-Solve.

---



**1-30** 

**Solution** We are to solve a system of 3 equations with 3 unknowns using EES.

***Analysis*** Using EES software, copy the following lines and paste on a blank EES screen to verify the solution:

```
2*x-y+z=5
3*x^2+2*y=z+2
x*y+2*z=8
```

*Answers*: ***x*** **= 1.141,** ***y*** **= 0.8159,** ***z*** **= 3.535.**

***Discussion*** To obtain the solution in EES, click on the icon that looks like a calculator, or Calculate-Solve.

---

**1-31** 

**Solution** We are to solve a system of 3 equations with 3 unknowns using EES.

***Analysis*** Using EES software, copy the following lines and paste on a blank EES screen to verify the solution:

```
x^2*y-z=1
x-3*y^0.5+x*z=-2
x+y-z=2
```

*Answers*: ***x*** **= 1,** ***y*** **= 1,** ***z*** **= 0.**

***Discussion*** To obtain the solution in EES, click on the icon that looks like a calculator, or Calculate-Solve.

---

## Review Problems

**1-32**

**Solution** The gravitational acceleration changes with altitude. Accounting for this variation, the weights of a body at different locations are to be determined.

***Analysis*** The weight of an 80-kg man at various locations is obtained by substituting the altitude *z* (values in m) into the relation

$$W = mg = (80\text{ kg})(9.807 - 3.32\times10^{-6}\,z\,\text{m/s}^2)\left(\frac{1\text{N}}{1\text{ kg}\cdot\text{m/s}^2}\right)$$

Sea level: ($z$ = 0 m): $W = 80\times(9.807\text{-}3.32\text{x}10^{-6}\times0) = 80\times9.807 =$ **784.6 N**

Denver: ($z$ = 1610 m): $W = 80\times(9.807\text{-}3.32\text{x}10^{-6}\times1610) = 80\times9.802 =$ **784.2 N**

Mt. Ev.: ($z$ = 8848 m): $W = 80\times(9.807\text{-}3.32\text{x}10^{-6}\times8848) = 80\times9.778 =$ **782.2 N**

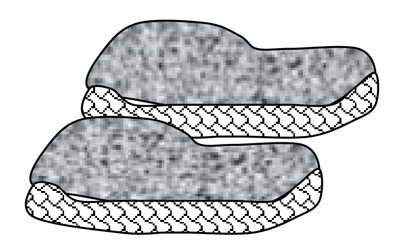
***Discussion*** We report 4 significant digits since the values are so close to each other. The percentage difference in weight from sea level to Mt. Everest is only about -0.3%, which is negligible for most engineering calculations.

**1-33**

**Solution** A man is considering buying a 12-oz steak for \$3.15, or a 320-g steak for \$2.80. The steak that is a better buy is to be determined.

***Assumptions*** The steaks are of identical quality.

***Analysis*** To make a comparison possible, we need to express the cost of each steak on a common basis. We choose 1 kg as the basis for comparison. Using proper conversion factors, the unit cost of each steak is determined to be



*12 ounce steak*: $\text{Unit Cost} = \left(\frac{\$3.15}{12\text{ oz}}\right)\left(\frac{16\text{ oz}}{1\text{ lbm}}\right)\left(\frac{1\text{ lbm}}{0.45359\text{ kg}}\right) = \mathbf{\$9.26/kg}$

*320 gram steak*:

$$\text{Unit Cost} = \left(\frac{\$2.80}{320\text{ g}}\right)\left(\frac{1000\text{ g}}{1\text{ kg}}\right) = \mathbf{\$8.75/kg}$$

Therefore, **the steak at the international market is a better buy**.

***Discussion*** Notice the unity conversion factors in the above equations.

---

**1-34**

**Solution** The thrust developed by the jet engine of a Boeing 777 is given to be 85,000 pounds. This thrust is to be expressed in N and kgf.

***Analysis*** Noting that 1 lbf = 4.448 N and 1 kgf = 9.81 N, the thrust developed is expressed in two other units as



Thrust in N: $\text{Thrust} = (85{,}000\text{ lbf})\left(\frac{4.448\text{ N}}{1\text{ lbf}}\right) = \mathbf{3.78\times10^5\ N}$

Thrust in kgf: $\text{Thrust} = (37.8\times10^5\text{ N})\left(\frac{1\text{ kgf}}{9.81\text{ N}}\right) = \mathbf{3.85\times10^4\ kgf}$

***Discussion*** Because the gravitational acceleration on earth is close to 10 $m/s^2$, it turns out that the two force units N and kgf differ by nearly a factor of 10. This can lead to confusion, and we recommend that you do not use the unit kgf.

---

## Design and Essay Problem

---

**1-35**

**Solution** We are to write an essay on mass- and volume-measurement devices.

***Discussion*** Students' essays should be unique and will differ from each other.

---



# Solutions Manual for

# Fluid Mechanics: Fundamentals and Applications

# by Çengel & Cimbala

---

# CHAPTER 2
# PROPERTIES OF FLUIDS

---

## Density and Specific Gravity

**2-1C**

**Solution** We are to discuss the difference between intensive and extensive properties.

***Analysis*** *Intensive properties* **do *not* depend on the size (extent) of the system** but *extensive properties* ***do*** **depend on the size (extent) of the system**.

***Discussion*** An example of an intensive property is temperature. An example of an extensive property is mass.

---

**2-2C**

**Solution** We are to define specific gravity and discuss its relationship to density.

***Analysis*** The *specific gravity***,** or *relative density***,** is defined as **the ratio of the density of a substance to the density of some standard substance at a specified temperature** (the standard is water at 4°C, for which $\rho_{H2O}$ = 1000 kg/m$^3$). That is, $\boxed{SG = \rho / \rho_{H2O}}$. When specific gravity is known, density is determined from $\boxed{\rho = SG \times \rho_{H2O}}$.

***Discussion*** Specific gravity is dimensionless and unitless [it is just a number without dimensions or units].

---

**2-3C**

**Solution** We are to discuss the applicability of the ideal gas law.

***Analysis*** A gas can be treated as an *ideal gas* when it is **at a *high temperature* and/or a *low pressure* relative to its critical temperature and pressure**.

***Discussion*** Air and many other gases at room temperature and pressure can be approximated as ideal gases without any significant loss of accuracy.

---

**2-4C**

**Solution** We are to discuss the difference between $R$ and $R_u$.

***Analysis*** $R_u$ is the ***universal gas constant*** that is the **same for all gases**, whereas $R$ is the ***specific gas constant*** that is **different for different gases**. These two are related to each other by $\boxed{R = R_u / M}$, where $M$ is the *molar mass* (also called the *molecular weight*) of the gas.

***Discussion*** Since molar mass has dimensions of mass per mole, $R$ and $R_u$ do not have the same dimensions or units.

---

**2-5**

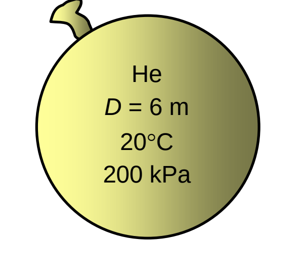
**Solution** A balloon is filled with helium gas. The number of moles and the mass of helium are to be determined.

***Assumptions*** At specified conditions, helium behaves as an ideal gas.

***Properties*** The universal gas constant is $R_u$ = 8.314 kPa.m$^3$/kmol.K. The molar mass of helium is 4.0 kg/kmol.

***Analysis*** The volume of the sphere is

$$V = \frac{4}{3}\pi r^3 = \frac{4}{3}\pi (3\text{ m})^3 = 113.1\text{ m}^3$$




Assuming ideal gas behavior, the number of moles of He is determined from

$$N = \frac{PV}{R_u T} = \frac{(200\text{ kPa})(113.1\text{ m}^3)}{(8.314\text{ kPa}\cdot\text{m}^3/\text{kmol}\cdot\text{K})(293\text{ K})} = \mathbf{9.286\text{ kmol}}$$

Then the mass of He is determined from

$$m = NM = (9.286\text{ kmol})(4.0\text{ kg/kmol}) = \mathbf{37.1\text{ kg}}$$

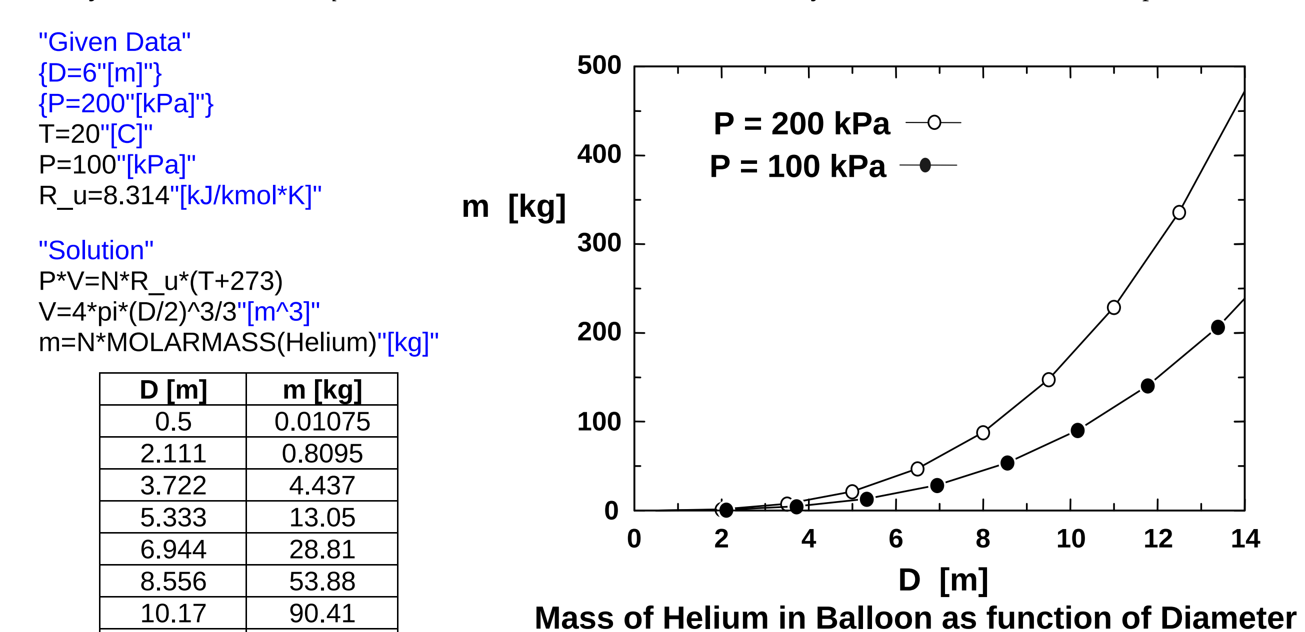
***Discussion*** Although the helium mass may seem large (about half the mass of an adult man!), it is much smaller than that of the air it displaces, and that is why helium balloons rise in the air.

---

**2-6** 


**Solution** A balloon is filled with helium gas. The effect of the balloon diameter on the mass of helium is to be investigated, and the results are to be tabulated and plotted.

***Analysis*** The EES *Equations* window is shown below, followed by the *Solution* window and the parametric table.



```
"Given Data"
{D=6"[m]"}
{P=200"[kPa]"}
T=20"[C]"
P=100"[kPa]"
R_u=8.314"[kJ/kmol*K]"

"Solution"
P*V=N*R_u*(T+273)
V=4*pi*(D/2)^3/3"[m^3]"
m=N*MOLARMASS(Helium)"[kg]"
```

| D [m] | m [kg] |
|---|---|
| 0.5 | 0.01075 |
| 2.111 | 0.8095 |
| 3.722 | 4.437 |
| 5.333 | 13.05 |
| 6.944 | 28.81 |
| 8.556 | 53.88 |
| 10.17 | 90.41 |
| 11.78 | 140.6 |
| 13.39 | 206.5 |
| 15 | 290.4 |



***Discussion*** Mass increases with diameter as expected, but not linearly since volume is proportional to $D^3$.

---

**2-7**

**Solution** An automobile tire is inflated with air. The pressure rise of air in the tire when the tire is heated and the amount of air that must be bled off to reduce the temperature to the original value are to be determined.

***Assumptions*** **1** At specified conditions, air behaves as an ideal gas. **2** The volume of the tire remains constant.

***Properties*** The gas constant of air is $R = 0.287$ kPa·m³/kg·K.

***Analysis*** Initially, the absolute pressure in the tire is

$$P_1 = P_g + P_{atm} = 210 + 100 = 310 \text{ kPa}$$

Treating air as an ideal gas and assuming the volume of the tire to remain constant, the final pressure in the tire is determined from

$$\frac{P_1V_1}{T_1} = \frac{P_2V_2}{T_2} \longrightarrow P_2 = \frac{T_2}{T_1}P_1 = \frac{323\text{K}}{298\text{K}}(310\text{kPa}) = 336\text{kPa}$$

Tire
25°C
210 kPa

Thus the pressure rise is

$$\Delta P = P_2 - P_1 = 336 - 310 = \mathbf{26.0\ kPa}$$

The amount of air that needs to be bled off to restore pressure to its original value is

$$m_1 = \frac{P_1V}{RT_1} = \frac{(310\text{kPa})(0.025\text{m}^3)}{(0.287\text{kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(298\text{K})} = 0.0906\text{kg}$$

$$m_2 = \frac{P_2V}{RT_2} = \frac{(310\text{kPa})(0.025\text{m}^3)}{(0.287\text{kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(323\text{K})} = 0.0836\text{kg}$$

$$\Delta m = m_1 - m_2 = 0.0906 - 0.0836 = \mathbf{0.0070\ kg}$$

***Discussion*** Notice that *absolute* rather than gage pressure must be used in calculations with the ideal gas law.

---

**2-8E**

**Solution** An automobile tire is under-inflated with air. The amount of air that needs to be added to the tire to raise its pressure to the recommended value is to be determined.

***Assumptions*** **1** At specified conditions, air behaves as an ideal gas. **2** The volume of the tire remains constant.

***Properties*** The gas constant of air is $R = 0.3704$ psia·ft³/lbm·R.

***Analysis*** The initial and final absolute pressures in the tire are

Tire
0.53 ft³
90°F
20 psia

$$P_1 = P_{g1} + P_{atm} = 20 + 14.6 = 34.6 \text{ psia}$$

$$P_2 = P_{g2} + P_{atm} = 30 + 14.6 = 44.6 \text{ psia}$$

Treating air as an ideal gas, the initial mass in the tire is

$$m_1 = \frac{P_1V}{RT_1} = \frac{(34.6\text{ psia})(0.53\text{ ft}^3)}{(0.3704\text{ psia}\cdot\text{ft}^3/\text{lbm}\cdot\text{R})(550\text{ R})} = 0.0900\text{ lbm}$$

Noting that the temperature and the volume of the tire remain constant, the final mass in the tire becomes

$$m_2 = \frac{P_2V}{RT_2} = \frac{(44.6\text{ psia})(0.53\text{ ft}^3)}{(0.3704\text{ psia}\cdot\text{ft}^3/\text{lbm}\cdot\text{R})(550\text{ R})} = 0.1160\text{ lbm}$$

Thus the amount of air that needs to be added is $\Delta m = m_2 - m_1 = 0.1160 - 0.0900 = \mathbf{0.0260\ lbm}$

***Discussion*** Notice that *absolute* rather than gage pressure must be used in calculations with the ideal gas law.

---

**2-9E**

**Solution** A rigid tank contains slightly pressurized air. The amount of air that needs to be added to the tank to raise its pressure and temperature to the recommended values is to be determined.

***Assumptions*** **1** At specified conditions, air behaves as an ideal gas. **2** The volume of the tank remains constant.

***Properties*** The gas constant of air is $R$ = 0.3704 psia·ft$^3$/lbm·R.

***Analysis*** Treating air as an ideal gas, the initial volume and the final mass in the tank are determined to be

$$V = \frac{m_1 R T_1}{P_1} = \frac{(20 \text{ lbm})(0.3704 \text{ psia}\cdot\text{ft}^3/\text{lbm}\cdot\text{R})(530 \text{ R})}{20 \text{ psia}} = 196.3 \text{ ft}^3$$

$$m_2 = \frac{P_2 V}{R T_2} = \frac{(35 \text{ psia})(196.3 \text{ ft}^3)}{(0.3704 \text{ psia}\cdot\text{ft}^3/\text{lbm}\cdot\text{R})(550 \text{ R})} = 33.73 \text{ lbm}$$

Air, 20 lbm
20 psia
70°F

Thus the amount of air added is

$$\Delta m = m_2 - m_1 = 33.73 - 20.0 = \mathbf{13.7 \text{ lbm}}$$

***Discussion*** As the temperature slowly decreases due to heat transfer, the pressure will also decrease.

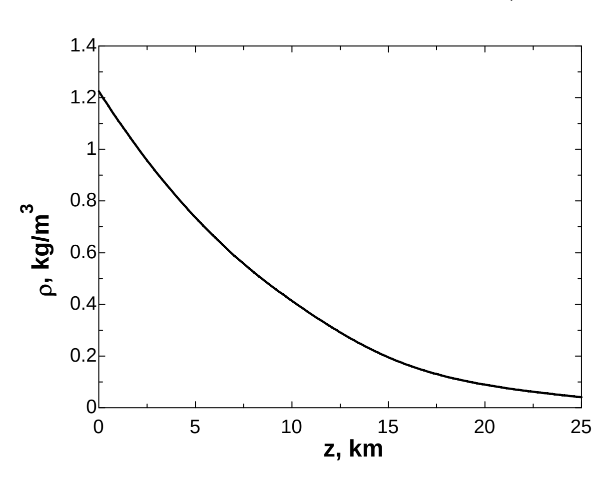
**2-10** 

**Solution** A relation for the variation of density with elevation is to be obtained, the density at 7 km elevation is to be calculated, and the mass of the atmosphere using the correlation is to be estimated.

***Assumptions*** **1** Atmospheric air behaves as an ideal gas. **2** The earth is perfectly spherical with a radius of 6377 km at sea level, and the thickness of the atmosphere is 25 km.

***Properties*** The density data are given in tabular form as a function of radius and elevation, where $r = z + 6377$ km:

| $r$, km | $z$, km | $\rho$, kg/m$^3$ |
|---|---|---|
| 6377 | 0 | 1.225 |
| 6378 | 1 | 1.112 |
| 6379 | 2 | 1.007 |
| 6380 | 3 | 0.9093 |
| 6381 | 4 | 0.8194 |
| 6382 | 5 | 0.7364 |
| 6383 | 6 | 0.6601 |
| 6385 | 8 | 0.5258 |
| 6387 | 10 | 0.4135 |
| 6392 | 15 | 0.1948 |
| 6397 | 20 | 0.08891 |
| 6402 | 25 | 0.04008 |




***Analysis*** Using EES, (1) Define a trivial function "rho= a+z" in the Equation window, (2) select new parametric table from Tables, and type the data in a two-column table, (3) select Plot and plot the data, and (4) select Plot and click on curve fit to get curve fit window. Then specify 2$^{nd}$ order polynomial and enter/edit equation. The results are:

$\rho(z) = a + bz + cz^2 = 1.20252 - 0.101674z + 0.0022375z^2$ for the unit of kg/m$^3$,
(or, $\rho(z) = (1.20252 - 0.101674z + 0.0022375z^2)\times 10^9$ for the unit of kg/km$^3$)

where $z$ is the vertical distance from the earth surface at sea level. At $z = 7$ km, the equation gives $\boldsymbol{\rho}$ **= 0.600 kg/m$^3$**.

($b$) The mass of atmosphere is evaluated by integration to be

$$m = \int_V \rho dV = \int_{z=0}^{h} (a + bz + cz^2)4\pi(r_0 + z)^2 dz = 4\pi\int_{z=0}^{h} (a + bz + cz^2)(r_0^2 + 2r_0 z + z^2)dz$$
$$= 4\pi\left[ar_0^2 h + r_0(2a + br_0)h^2/2 + (a + 2br_0 + cr_0^2)h^3/3 + (b + 2cr_0)h^4/4 + ch^5/5\right]$$

where $r_0 = 6377$ km is the radius of the earth, $h = 25$ km is the thickness of the atmosphere. Also, $a = 1.20252$, $b = -0.101674$, and $c = 0.0022375$ are the constants in the density function. Substituting and multiplying by the factor $10^9$ to convert the density from units of kg/km$^3$ to kg/m$^3$, the mass of the atmosphere is determined to be approximately

$m$ = **5.09×10$^{18}$ kg**

EES Solution for final result:

```
a=1.2025166
b=-0.10167
c=0.0022375
r=6377
h=25
m=4*pi*(a*r^2*h+r*(2*a+b*r)*h^2/2+(a+2*b*r+c*r^2)*h^3/3+(b+2*c*r)*h^4/4+c*h^5/5)*1E+9
```

***Discussion*** At 7 km, the density of the air is approximately half of its value at sea level.

## Vapor Pressure and Cavitation

### 2-11C

**Solution** We are to define vapor pressure and discuss its relationship to saturation pressure.

***Analysis*** The *vapor pressure* $P_v$ of a pure substance is defined as **the pressure exerted by a vapor in phase equilibrium with its liquid at a given temperature**. In general, the pressure of a vapor or gas, whether it exists alone or in a mixture with other gases, is called the *partial pressure*. During phase change processes between the liquid and vapor phases of a pure substance, the **saturation pressure and the vapor pressure are equivalent** since the vapor is pure.

***Discussion*** Partial pressure is not necessarily equal to vapor pressure. For example, on a dry day (low relative humidity), the partial pressure of water vapor in the air is less than the vapor pressure of water. If, however, the relative humidity is 100%, the partial pressure and the vapor pressure are equal.

### 2-12C

**Solution** We are to discuss whether the boiling temperature of water increases as pressure increases.

***Analysis*** **Yes**. The saturation temperature of a pure substance depends on pressure; in fact, it *increases* with pressure. The higher the pressure, the higher the saturation or boiling temperature.

***Discussion*** This fact is easily seen by looking at the saturated water property tables. Note that boiling temperature and saturation pressure at a given pressure are equivalent.

### 2-13C

**Solution** We are to determine if temperature increases or remains constant when the pressure of a boiling substance increases.

***Analysis*** If the pressure of a substance increases during a boiling process, **the temperature also increases** since the boiling (or saturation) temperature of a pure substance depends on pressure and increases with it.

***Discussion*** We are assuming that the liquid will continue to boil. If the pressure is increased fast enough, boiling may stop until the temperature has time to reach its new (higher) boiling temperature. A pressure cooker uses this principle.

### 2-14C

**Solution** We are to define and discuss cavitation.

***Analysis*** In the flow of a liquid, *cavitation* is the **vaporization that may occur at locations where the pressure drops below the vapor pressure**. The vapor bubbles collapse as they are swept away from the low pressure regions, generating highly destructive, extremely high-pressure waves. This phenomenon is a common cause for drop in performance and even the erosion of impeller blades.

***Discussion*** The word "cavitation" comes from the fact that a vapor bubble or "cavity" appears in the liquid. Not all cavitation is undesirable. It turns out that some underwater vehicles employ "super cavitation" on purpose to reduce drag.

**2-15**

**Solution** The minimum pressure in a piping system to avoid cavitation is to be determined.

***Properties*** The vapor pressure of water at 40°C is 7.38 kPa.

***Analysis*** To avoid cavitation, the pressure anywhere in the flow should not be allowed to drop below the vapor (or saturation) pressure at the given temperature. That is,

$$P_{\min} = P_{\text{sat @ 40°C}} = \mathbf{7.38\ kPa}$$

Therefore, the pressure should be maintained above 7.38 kPa everywhere in flow.

***Discussion*** Note that the vapor pressure increases with increasing temperature, and thus the risk of cavitation is greater at higher fluid temperatures.

**2-16**

**Solution** The minimum pressure in a pump is given. It is to be determined if there is a danger of cavitation.

***Properties*** The vapor pressure of water at 20°C is 2.339 kPa.

***Analysis*** To avoid cavitation, the pressure everywhere in the flow should remain above the vapor (or saturation) pressure at the given temperature, which is

$$P_v = P_{\text{sat @ 20°C}} = 2.339\text{ kPa}$$

The minimum pressure in the pump is 2 kPa, which is less than the vapor pressure. Therefore, a **there is danger of cavitation in the pump**.

***Discussion*** Note that the vapor pressure increases with increasing temperature, and thus there is a greater danger of cavitation at higher fluid temperatures.

**2-17E**

**Solution** The minimum pressure in a pump is given. It is to be determined if there is a danger of cavitation.

***Properties*** The vapor pressure of water at 70°F is 0.3632 psia.

***Analysis*** To avoid cavitation, the pressure everywhere in the flow should remain above the vapor (or saturation) pressure at the given temperature, which is

$$P_v = P_{\text{sat @ 70°F}} = 0.3632\text{ psia}$$

The minimum pressure in the pump is 0.1 psia, which is less than the vapor pressure. Therefore, **there is danger of cavitation in the pump**.

***Discussion*** Note that the vapor pressure increases with increasing temperature, and the danger of cavitation increases at higher fluid temperatures.

**2-18**

**Solution** The minimum pressure in a pump to avoid cavitation is to be determined.

***Properties*** The vapor pressure of water at 25°C is 3.17 kPa.

***Analysis*** To avoid cavitation, the pressure anywhere in the system should not be allowed to drop below the vapor (or saturation) pressure at the given temperature. That is,

$$P_{\min} = P_{\text{sat @ 25°C}} = \mathbf{3.17\ kPa}$$

Therefore, the lowest pressure that can exist in the pump is 3.17 kPa.

***Discussion*** Note that the vapor pressure increases with increasing temperature, and thus the risk of cavitation is greater at higher fluid temperatures.

## Energy and Specific Heats

### 2-19C

**Solution** We are to discuss the difference between macroscopic and microscopic forms of energy.

***Analysis*** The *macroscopic* forms of energy are those **a system possesses as a whole** with respect to some outside reference frame. The *microscopic* forms of energy, on the other hand, are those **related to the molecular structure of a system and the degree of the molecular activity**, and are independent of outside reference frames.

***Discussion*** We mostly deal with macroscopic forms of energy in fluid mechanics.

### 2-20C

**Solution** We are to define total energy and identify its constituents.

***Analysis*** The **sum of all forms of the energy a system possesses** is called *total energy*. In the absence of magnetic, electrical, and surface tension effects, **the total energy of a system consists of the kinetic, potential, and internal energies**.

***Discussion*** All three constituents of total energy (kinetic, potential, and internal) need to be considered in an analysis of a general fluid flow.

### 2-21C

**Solution** We are to list the forms of energy that contribute to the internal energy of a system.

***Analysis*** The *internal energy* of a system is made up of **sensible, latent, chemical, and nuclear energies**. The sensible internal energy is due to translational, rotational, and vibrational effects.

***Discussion*** We deal with the flow of a single phase fluid in most problems in this textbook; therefore, latent, chemical, and nuclear energies do not need to be considered.

### 2-22C

**Solution** We are to discuss the relationship between heat, internal energy, and thermal energy.

***Analysis*** *Thermal energy* is the **sensible and latent forms of internal energy**. It does not include chemical or nuclear forms of energy. In common terminology, thermal energy is referred to as *heat*. However, like work, heat is not a property, whereas thermal energy is a property.

***Discussion*** Technically speaking, “heat” is defined only when there is *heat transfer*, whereas the energy state of a substance can always be defined, even if no heat transfer is taking place.

### 2-23C

**Solution** We are to define and discuss flow energy.

***Analysis*** *Flow energy* or *flow work* is the **energy needed to push a fluid into or out of a control volume**. Fluids at rest do not possess any flow energy.

***Discussion*** Flow energy is not a fundamental quantity, like kinetic or potential energy. However, it is a useful concept in fluid mechanics since fluids are often forced into and out of control volumes in practice.

**2-24C**

**Solution** We are to compare the energies of flowing and non-flowing fluids.

***Analysis*** **A flowing fluid possesses *flow energy*, which is the energy needed to push a fluid into or out of a control volume**, in addition to the forms of energy possessed by a non-flowing fluid. The total energy of a non-flowing fluid consists of internal and potential energies. If the fluid is moving as a rigid body, but not *flowing*, it may also have kinetic energy (e.g., gasoline in a tank truck moving down the highway at constant speed with no sloshing). The total energy of a flowing fluid consists of internal, kinetic, potential, *and* flow energies.

***Discussion*** Flow energy is not to be confused with kinetic energy, even though both are zero when the fluid is at rest.

**2-25C**

**Solution** We are to explain how changes in internal energy can be determined.

***Analysis*** Using specific heat values at the average temperature, the changes in the specific internal energy of ideal gases can be determined from $\boxed{\Delta u = c_{v,avg}\Delta T}$. For incompressible substances, $c_p \cong c_v \cong c$ and $\boxed{\Delta u = c_{avg}\Delta T}$.

***Discussion*** If the fluid can be treated as neither incompressible nor an ideal gas, property tables must be used.

**2-26C**

**Solution** We are to explain how changes in enthalpy can be determined.

***Analysis*** Using specific heat values at the average temperature, the changes in specific enthalpy of ideal gases can be determined from $\boxed{\Delta h = c_{p,avg}\Delta T}$. For incompressible substances, $c_p \cong c_v \cong c$ and $\boxed{\Delta h = \Delta u + v\Delta P \cong c_{avg}\Delta T + v\Delta P}$.

***Discussion*** If the fluid can be treated as neither incompressible nor an ideal gas, property tables must be used.

## Coefficient of Compressibility

**2-27C**

**Solution** We are to discuss the coefficient of compressibility and the isothermal compressibility.

***Analysis*** The *coefficient of compressibility* represents the **variation of pressure of a fluid with volume or density at constant temperature**. *Isothermal compressibility* is the inverse of the coefficient of compressibility, and it represents the **fractional change in volume or density corresponding to a change in pressure**.

***Discussion*** The coefficient of compressibility of an ideal gas is equal to its absolute pressure.

**2-28C**

**Solution** We are to define the coefficient of volume expansion.

***Analysis*** The *coefficient of volume expansion* represents the **variation of the *density* of a fluid with temperature at constant pressure**. It differs from the coefficient of compressibility in that the latter represents the variation of *pressure* of a fluid with density at constant temperature.

***Discussion*** The coefficient of volume expansion of an ideal gas is equal to the inverse of its absolute temperature.

**2-29C**

**Solution** We are to discuss the sign of the coefficient of compressibility and the coefficient of volume expansion.

***Analysis*** The *coefficient of compressibility* of a fluid **cannot be negative**, but the *coefficient of volume expansion* **can be negative** (e.g., liquid water below 4°C).

***Discussion*** This is the reason that ice floats on water.

---

**2-30**

**Solution** The percent increase in the density of an ideal gas is given for a moderate pressure. The percent increase in density of the gas when compressed at a higher pressure is to be determined.

***Assumptions*** The gas behaves an ideal gas.

***Analysis*** For an ideal gas, $P = \rho RT$ and $(\partial P/\partial \rho)_T = RT = P/\rho$ , and thus $\kappa_{\text{ideal gas}} = P$ . Therefore, the coefficient of compressibility of an ideal gas is equal to its absolute pressure, and the coefficient of compressibility of the gas increases with increasing pressure.

Substituting $\kappa = P$ into the definition of the coefficient of compressibility $\kappa \cong -\frac{\Delta P}{\Delta v/v} \cong \frac{\Delta P}{\Delta \rho/\rho}$ and rearranging gives

$$\frac{\Delta \rho}{\rho} = \frac{\Delta P}{P}$$

Therefore, **the percent increase of density of an ideal gas during isothermal compression is equal to the percent increase in pressure**.

At 10 atm: $\frac{\Delta \rho}{\rho} = \frac{\Delta P}{P} = \frac{11-10}{10} = 10\%$

At 100 atm: $\frac{\Delta \rho}{\rho} = \frac{\Delta P}{P} = \frac{101-100}{100} = 1\%$

Therefore, a pressure change of 1 atm causes a density change of 10% at 10 atm and a density change of 1% at 100 atm.

***Discussion*** If temperature were also allowed to change, the relationship would not be so simple.

---

**2-31**

**Solution** Using the definition of the coefficient of volume expansion and the expression $\beta_{\text{ideal gas}} = 1/T$ , it is to be shown that the percent increase in the specific volume of an ideal gas during isobaric expansion is equal to the percent increase in absolute temperature.

***Assumptions*** The gas behaves an ideal gas.

***Analysis*** The coefficient of volume expansion $\beta$ can be expressed as $\beta = \frac{1}{v}\left(\frac{\partial v}{\partial T}\right)_P \approx \frac{\Delta v/v}{\Delta T}$ .

Noting that $\beta_{\text{ideal gas}} = 1/T$ for an ideal gas and rearranging give

$$\frac{\Delta v}{v} = \frac{\Delta T}{T}$$

Therefore, **the percent increase in the specific volume of an ideal gas during isobaric expansion is equal to the percent increase in absolute temperature**.

***Discussion*** We must be careful to use *absolute* temperature (K or R), not relative temperature ($^o$C or $^o$F).

---

**2-32**

**Solution** Water at a given temperature and pressure is compressed to a high pressure isothermally. The increase in the density of water is to be determined.

***Assumptions*** **1** The isothermal compressibility is constant in the given pressure range. **2** An approximate analysis is performed by replacing differential changes by finite changes.

***Properties*** The density of water at 20°C and 1 atm pressure is $\rho_1$ = 998 kg/m$^3$. The isothermal compressibility of water is given to be $\alpha$ = 4.80 × 10$^{-5}$ atm$^{-1}$.

***Analysis*** When differential quantities are replaced by differences and the properties $\alpha$ and $\beta$ are assumed to be constant, the change in density in terms of the changes in pressure and temperature is expressed approximately as

$$\Delta\rho = \alpha\rho\Delta P - \beta\rho\Delta T$$

The change in density due to a change of pressure from 1 atm to 800 atm at constant temperature is

$$\Delta\rho = \alpha\rho\Delta P = (4.80\times10^{-5}\text{ atm}^{-1})(998\text{ kg/m}^3)(800-1)\text{atm} = \mathbf{38.3\ kg/m^3}$$

***Discussion*** Note that the density of water increases from 998 to 1036.3 kg/m$^3$ while being compressed, as expected. This problem can be solved more accurately using differential analysis when functional forms of properties are available.

---

**2-33**

**Solution** Water at a given temperature and pressure is heated to a higher temperature at constant pressure. The change in the density of water is to be determined.

***Assumptions*** **1** The coefficient of volume expansion is constant in the given temperature range. **2** An approximate analysis is performed by replacing differential changes in quantities by finite changes.

***Properties*** The density of water at 15°C and 1 atm pressure is $\rho_1$ = 999.1 kg/m$^3$. The coefficient of volume expansion at the average temperature of (15+95)/2 = 55°C is $\beta$ = 0.484 × 10$^{-3}$ K$^{-1}$.

***Analysis*** When differential quantities are replaced by differences and the properties $\alpha$ and $\beta$ are assumed to be constant, the change in density in terms of the changes in pressure and temperature is expressed approximately as

$$\Delta\rho = \alpha\rho\Delta P - \beta\rho\Delta T$$

The change in density due to the change of temperature from 15°C to 95°C at constant pressure is

$$\Delta\rho = -\beta\rho\Delta T = -(0.484\times10^{-3}\text{ K}^{-1})(999.1\text{ kg/m}^3)(95-15)\text{K} = \mathbf{-38.7\ kg/m^3}$$

***Discussion*** Noting that $\Delta\rho = \rho_2 - \rho_1$, the density of water at 95°C and 1 atm is

$$\rho_2 = \rho_1 + \Delta\rho = 999.1 + (-38.7) = 960.4\text{ kg/m}^3$$

which is very close to the listed value of 961.5 kg/m$^3$ at 95°C in water table in the Appendix. This is mostly due to $\beta$ varying with temperature almost linearly. Note that the density of water decreases while being heated, as expected. This problem can be solved more accurately using differential analysis when functional forms of properties are available.

---

**2-34**

**Solution** Saturated refrigerant-134a at a given temperature is cooled at constant pressure. The change in the density of the refrigerant is to be determined.

***Assumptions*** **1** The coefficient of volume expansion is constant in the given temperature range. **2** An approximate analysis is performed by replacing differential changes in quantities by finite changes.

***Properties*** The density of saturated liquid R-134a at 10°C is $\rho_1$ =1261 kg/m$^3$. The coefficient of volume expansion at the average temperature of (10+0)/2 = 5°C is $\beta$ = 0.00269 K$^{-1}$.

***Analysis*** When differential quantities are replaced by differences and the properties $\alpha$ and $\beta$ are assumed to be constant, the change in density in terms of the changes in pressure and temperature is expressed approximately as

$$\Delta\rho = \alpha\rho\Delta P - \beta\rho\Delta T$$

The change in density due to the change of temperature from 10°C to 0°C at constant pressure is

$$\Delta\rho = -\beta\rho\Delta T = -(0.00269\,\text{K}^{-1})(1261\,\text{kg/m}^3)(0-10)\text{K} = \mathbf{33.9\ kg/m^3}$$

***Discussion*** Noting that $\Delta\rho = \rho_2 - \rho_1$, the density of R-134a at 0°C is

$$\rho_2 = \rho_1 + \Delta\rho = 1261 + 33.9 = 1294.9\ \text{kg/m}^3$$

which is almost identical to the listed value of 1295 kg/m$^3$ at 0°C in R-134a table in the Appendix. This is mostly due to $\beta$ varying with temperature almost linearly. Note that the density increases during cooling, as expected.

---

**2-35**

**Solution** A water tank completely filled with water can withstand tension caused by a volume expansion of 2%. The maximum temperature rise allowed in the tank without jeopardizing safety is to be determined.

***Assumptions*** **1** The coefficient of volume expansion is constant. **2** An approximate analysis is performed by replacing differential changes in quantities by finite changes. **3** The effect of pressure is disregarded.

***Properties*** The average volume expansion coefficient is given to be $\beta$ = 0.377 × 10$^{-3}$ K$^{-1}$.

***Analysis*** When differential quantities are replaced by differences and the properties $\alpha$ and $\beta$ are assumed to be constant, the change in density in terms of the changes in pressure and temperature is expressed approximately as

$$\Delta\rho = \alpha\rho\Delta P - \beta\rho\Delta T$$

A volume increase of 2% corresponds to a density decrease of 2%, which can be expressed as $\Delta\rho = -0.02\rho$. Then the decrease in density due to a temperature rise of $\Delta T$ at constant pressure is

$$-0.02\rho = -\beta\rho\Delta T$$

Solving for $\Delta T$ and substituting, the maximum temperature rise is determined to be

$$\Delta T = \frac{0.02}{\beta} = \frac{0.02}{0.377\times10^{-3}\ \text{K}^{-1}} = \mathbf{53.0\ K} = \mathbf{53.0^\circ C}$$

***Discussion*** This result is conservative since in reality the increasing pressure will tend to compress the water and increase its density.

---

**2-36**

**Solution** A water tank completely filled with water can withstand tension caused by a volume expansion of 1%. The maximum temperature rise allowed in the tank without jeopardizing safety is to be determined.

***Assumptions*** **1** The coefficient of volume expansion is constant. **2** An approximate analysis is performed by replacing differential changes in quantities by finite changes. **3** The effect of pressure is disregarded.

***Properties*** The average volume expansion coefficient is given to be $\beta = 0.377 \times 10^{-3}\ \text{K}^{-1}$.

***Analysis*** When differential quantities are replaced by differences and the properties $\alpha$ and $\beta$ are assumed to be constant, the change in density in terms of the changes in pressure and temperature is expressed approximately as

$$\Delta\rho = \alpha\rho\Delta P - \beta\rho\Delta T$$

A volume increase of 1% corresponds to a density decrease of 1%, which can be expressed as $\Delta\rho = -0.01\rho$. Then the decrease in density due to a temperature rise of $\Delta T$ at constant pressure is

$$-0.01\rho = -\beta\rho\Delta T$$

Solving for $\Delta T$ and substituting, the maximum temperature rise is determined to be

$$\Delta T = \frac{0.01}{\beta} = \frac{0.01}{0.377 \times 10^{-3}\ \text{K}^{-1}} = \mathbf{26.5\ K} = \mathbf{26.5^\circ C}$$

***Discussion*** This result is conservative since in reality the increasing pressure will tend to compress the water and increase its density. The change in temperature is exactly half of that of the previous problem, as expected.

---

**2-37**

**Solution** The density of seawater at the free surface and the bulk modulus of elasticity are given. The density and pressure at a depth of 2500 m are to be determined.

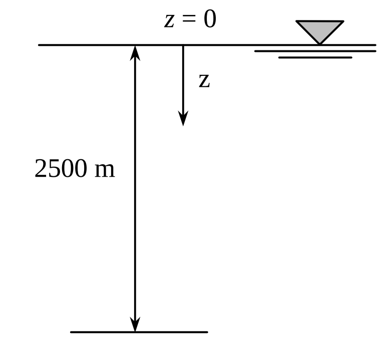
***Assumptions*** **1** The temperature and the bulk modulus of elasticity of seawater is constant. **2** The gravitational acceleration remains constant.

***Properties*** The density of seawater at free surface where the pressure is given to be 1030 kg/m$^3$, and the bulk modulus of elasticity of seawater is given to be 2.34 × 10$^9$ N/m$^2$.

***Analysis*** The coefficient of compressibility or the bulk modulus of elasticity of fluids is expressed as

$$\kappa = \rho\left(\frac{\partial P}{\partial \rho}\right)_T \quad \text{or} \quad \kappa = \rho\frac{dP}{d\rho} \quad \text{(at constant } T\text{)}$$

The differential pressure change across a differential fluid height of $dz$ is given as




$$dP = \rho g dz$$

Combining the two relations above and rearranging,

$$\kappa = \rho\frac{\rho g dz}{d\rho} = g\rho^2\frac{dz}{d\rho} \quad \rightarrow \quad \frac{d\rho}{\rho^2} = \frac{gdz}{\kappa}$$

Integrating from $z$ = 0 where $\rho = \rho_0 = 1030\ \text{kg/m}^3$ to $z = z$ where $\rho = \rho$ gives

$$\int_{\rho_0}^{\rho}\frac{d\rho}{\rho^2} = \frac{g}{\kappa}\int_0^z dz \quad \rightarrow \quad \frac{1}{\rho_0} - \frac{1}{\rho} = \frac{gz}{\kappa}$$

Solving for $\rho$ gives the variation of density with depth as

$$\rho = \frac{1}{(1/\rho_0) - (gz/\kappa)}$$

Substituting into the pressure change relation $dP = \rho g dz$ and integrating from $z$ = 0 where $P = P_0 = 98\ \text{kPa}$ to $z = z$ where $P = P$ gives

$$\int_{P_0}^{P} dP = \int_0^z \frac{gdz}{(1/\rho_0) - (gz/\kappa)} \quad \rightarrow \quad P = P_0 + \kappa \ln\left(\frac{1}{1 - (\rho_0 gz/\kappa)}\right)$$

which is the desired relation for the variation of pressure in seawater with depth. At $z$ = 2500 m, the values of density and pressure are determined by substitution to be

$$\rho = \frac{1}{1/(1030\ \text{kg/m}^3) - (9.81\ \text{m/s}^2)(2500\ \text{m})/(2.34\times10^9\ \text{N/m}^2)} = \mathbf{1041\ kg/m^3}$$

$$P = (98{,}000\ \text{Pa}) + (2.34\times10^9\ \text{N/m}^2)\ln\left(\frac{1}{1-(1030\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(2500\ \text{m})/(2.34\times10^9\ \text{N/m}^2)}\right)$$

$$= 2.550\times10^7\ \text{Pa}$$

$$= \mathbf{25.50\ MPa}$$

since 1 Pa = 1 N/m$^2$ = 1 kg/m·s$^2$ and 1 kPa = 1000 Pa.

***Discussion*** Note that if we assumed $\rho = \rho_o$ = constant at 1030 kg/m$^3$, the pressure at 2500 m would be $P = P_0 + \rho g z$ = 0.098 + 25.26 = 25.36 MPa. Then the density at 2500 m is estimated to be

$$\Delta\rho = \rho\alpha\Delta P = (1030)(2340\ \text{MPa})^{-1}(25.26\ \text{MPa}) = 11.1\ \text{kg/m}^3 \text{ and thus } \rho = 1041\ \text{kg/m}^3$$

## Viscosity

**2-38C**

**Solution** We are to define and discuss viscosity.

***Analysis*** *Viscosity* is **a measure of the "stickiness" or "resistance to deformation" of a fluid**. It is due to the internal frictional force that develops between different layers of fluids as they are forced to move relative to each other. Viscosity is caused by the cohesive forces between the molecules in liquids, and by the molecular collisions in gases. In general, **liquids have higher dynamic viscosities than gases**.

***Discussion*** The ratio of viscosity $\mu$ to density $\rho$ often appears in the equations of fluid mechanics, and is defined as the *kinematic viscosity*, $\nu = \mu/\rho$.

---

**2-39C**

**Solution** We are to discuss Newtonian fluids.

***Analysis*** **Fluids whose shear stress is linearly proportional to the velocity gradient** (shear strain) are called *Newtonian fluids*. Most common fluids such as water, air, gasoline, and oils are Newtonian fluids.

***Discussion*** In the differential analysis of fluid flow, only Newtonian fluids are considered in this textbook.

---

**2-40C**

**Solution** We are to compare the settling speed of balls dropped in water and oil; namely, we are to determine which will reach the bottom of the container first.

***Analysis*** When two identical small glass balls are dropped into two identical containers, one filled with water and the other with oil, **the ball dropped in water will reach the bottom of the container first** because of the much lower viscosity of water relative to oil.

***Discussion*** Oil is very viscous, with typical values of viscosity approximately 800 times greater than that of water at room temperature.

---

**2-41C**

**Solution** We are to discuss how dynamic viscosity varies with temperature in liquids and gases.

***Analysis*** (*a*) The **dynamic viscosity of liquids decreases with temperature**. (*b*) The **dynamic viscosity of gases increases with temperature**.

***Discussion*** A good way to remember this is that a car engine is much harder to start in the winter because the oil in the engine has a higher viscosity at low temperatures.

---

**2-42C**

**Solution** We are to discuss how kinematic viscosity varies with temperature in liquids and gases.

***Analysis*** (*a*) **For *liquids*, the kinematic viscosity decreases with temperature**. (*b*) **For *gases*, the kinematic viscosity increases with temperature**.

***Discussion*** You can easily verify this by looking at the appendices.

---

**2-43**

**Solution** A block is moved at constant velocity on an inclined surface. The force that needs to be applied in the horizontal direction when the block is dry, and the percent reduction in the required force when an oil film is applied on the surface are to be determined.

***Assumptions*** **1** The inclined surface is plane (perfectly flat, although tilted). **2** The friction coefficient and the oil film thickness are uniform. **3** The weight of the oil layer is negligible.

***Properties*** The absolute viscosity of oil is given to be $\mu = 0.012$ Pa·s = 0.012 N·s/m$^2$.

***Analysis*** (*a*) The velocity of the block is constant, and thus its acceleration and the net force acting on it are zero. A free body diagram of the block is given. Then the force balance gives

$$\sum F_x = 0: \quad F_1 - F_f \cos 20° - F_{N1} \sin 20° = 0 \quad (1)$$

$$\sum F_y = 0: \quad F_{N1} \cos 20° - F_f \sin 20° - W = 0 \quad (2)$$

Friction force: $F_f = fF_{N1}$ (3)

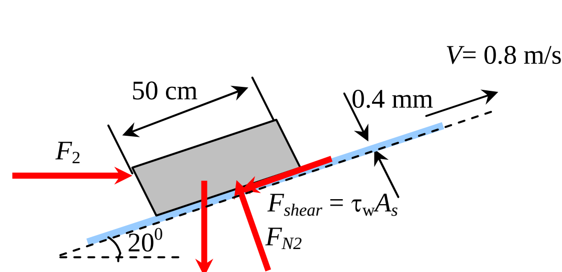
Substituting Eq. (3) into Eq. (2) and solving for $F_{N1}$ gives

$$F_{N1} = \frac{W}{\cos 20° - f \sin 20°} = \frac{150 \text{ N}}{\cos 20° - 0.27 \sin 20°} = 177.0 \text{ N}$$

Then from Eq. (1):

$$F_1 = F_f \cos 20° + F_{N1} \sin 20° = (0.27 \times 177 \text{ N}) \cos 20° + (177 \text{ N}) \sin 20° = \mathbf{105.5\ N}$$

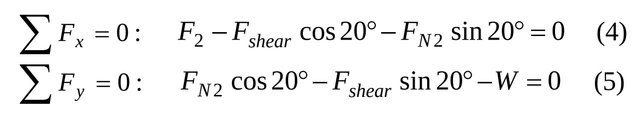
(*b*) In this case, the friction force is replaced by the shear force applied on the bottom surface of the block due to the oil. Because of the no-slip condition, the oil film sticks to the inclined surface at the bottom and the lower surface of the block at the top. Then the shear force is expressed as




$$F_{shear} = \tau_w A_s$$
$$= \mu A_s \frac{V}{h}$$
$$= \left(0.012 \text{ N}\cdot\text{s/m}^2\right)\left(0.5 \times 0.2 \text{ m}^2\right) \frac{0.8 \text{ m/s}}{4 \times 10^{-4} \text{ m}}$$
$$= 2.4 \text{ N}$$

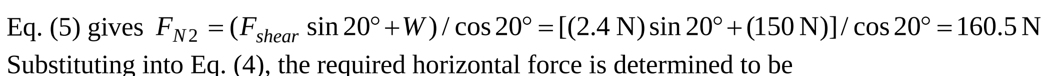


Replacing the friction force by the shear force in part (*a*),



$$\sum F_x = 0: \quad F_2 - F_{shear} \cos 20° - F_{N2} \sin 20° = 0 \quad (4)$$

$$\sum F_y = 0: \quad F_{N2} \cos 20° - F_{shear} \sin 20° - W = 0 \quad (5)$$



Eq. (5) gives $F_{N2} = (F_{shear} \sin 20° + W) / \cos 20° = [(2.4 \text{ N}) \sin 20° + (150 \text{ N})] / \cos 20° = 160.5 \text{ N}$

Substituting into Eq. (4), the required horizontal force is determined to be

$$F_2 = F_{shear} \cos 20° + F_{N2} \sin 20° = (2.4 \text{ N}) \cos 20° + (160.5 \text{ N}) \sin 20° = 57.2 \text{ N}$$

Then, our final result is expressed as

$$\text{Percentage reduction in required force} = \frac{F_1 - F_2}{F_1} \times 100\% = \frac{105.5 - 57.2}{105.5} \times 100\% = \mathbf{45.8\%}$$

***Discussion*** Note that the force required to push the block on the inclined surface reduces significantly by oiling the surface.

**2-44**

**Solution** The velocity profile of a fluid flowing though a circular pipe is given. The friction drag force exerted on the pipe by the fluid in the flow direction per unit length of the pipe is to be determined.

***Assumptions*** The viscosity of the fluid is constant.

***Analysis*** The wall shear stress is determined from its definition to be

$$\tau_w = -\mu \frac{du}{dr}\bigg|_{r=R} = -\mu u_{max} \frac{d}{dr}\left(1 - \frac{r^n}{R^n}\right)_{r=R} = -\mu u_{max} \frac{-nr^{n-1}}{R^n}\bigg|_{r=R} = \frac{n\mu u_{max}}{R}$$

Note that the quantity $du/dr$ is negative in pipe flow, and the negative sign is added to the $\tau_w$ relation for pipes to make shear stress in the positive (flow) direction a positive quantity. (Or, $du/dr = - du/dy$ since $y = R - r$). Then the friction drag force exerted by the fluid on the inner surface of the pipe becomes

$$F = \tau_w A_w = \frac{n\mu u_{max}}{R}(2\pi R)L = 2n\pi\mu u_{max} L$$



Therefore, the drag force per unit length of the pipe is

$$\boxed{F / L = 2n\pi\mu u_{max}}.$$

***Discussion*** Note that the drag force acting on the pipe in this case is independent of the pipe diameter.

**2-45**

**Solution** A thin flat plate is pulled horizontally through an oil layer sandwiched between two plates, one stationary and the other moving at a constant velocity. The location in oil where the velocity is zero and the force that needs to be applied on the plate are to be determined.

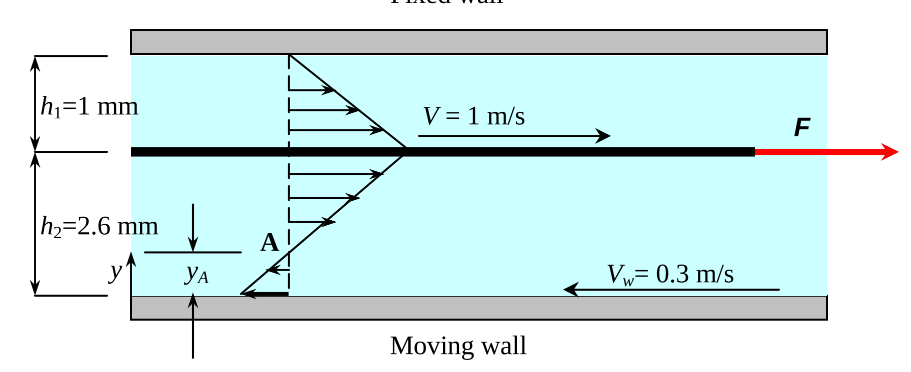
***Assumptions*** **1** The thickness of the plate is negligible. **2** The velocity profile in each oil layer is linear.

***Properties*** The absolute viscosity of oil is given to be $\mu$ = 0.027 Pa·s = 0.027 N·s/m$^2$.

***Analysis*** (*a*) The velocity profile in each oil layer relative to the fixed wall is as shown in the figure below. The point of zero velocity is indicated by point *A*, and its distance from the lower plate is determined from geometric considerations (the similarity of the two triangles in the lower oil layer) to be

$$\frac{2.6 - y_A}{y_A} = \frac{1}{0.3} \quad \rightarrow \quad y_A = \mathbf{0.60\ mm}$$




(*b*) The magnitudes of shear forces acting on the upper and lower surfaces of the plate are

$$F_{\text{shear, upper}} = \tau_{w,\text{upper}} A_s = \mu A_s \left|\frac{du}{dy}\right| = \mu A_s \frac{V-0}{h_1} = (0.027\ \text{N}\cdot\text{s/m}^2)(0.2\times 0.2\ \text{m}^2)\frac{1\ \text{m/s}}{1.0\times 10^{-3}\ \text{m}} = 1.08\ \text{N}$$

$$F_{\text{shear, lower}} = \tau_{w,\text{lower}} A_s = \mu A_s \left|\frac{du}{dy}\right| = \mu A_s \frac{V-V_w}{h_2} = (0.027\ \text{N}\cdot\text{s/m}^2)(0.2\times 0.2\ \text{m}^2)\frac{[1-(-0.3)]\ \text{m/s}}{2.6\times 10^{-3}\ \text{m}} = 0.54\ \text{N}$$

Noting that both shear forces are in the opposite direction of motion of the plate, the force *F* is determined from a force balance on the plate to be

$$F = F_{\text{shear,upper}} + F_{\text{shear,lower}} = 1.08 + 0.54 = \mathbf{1.62\ N}$$

***Discussion*** Note that wall shear is a friction force between a solid and a liquid, and it acts in the opposite direction of motion.

**2-46**

**Solution** A frustum shaped body is rotating at a constant angular speed in an oil container. The power required to maintain this motion and the reduction in the required power input when the oil temperature rises are to be determined.

***Assumptions*** The thickness of the oil layer remains constant.

***Properties*** The absolute viscosity of oil is given to be $\mu$ = 0.1 Pa·s = 0.1 N·s/m$^2$ at 20°C and 0.0078 Pa·s at 80°C.

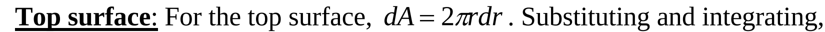
***Analysis*** The velocity gradient anywhere in the oil of film thickness $h$ is $V/h$ where $V = \omega r$ is the tangential velocity. Then the wall shear stress anywhere on the surface of the frustum at a distance $r$ from the axis of rotation is

$$\tau_w = \mu \frac{du}{dr} = \mu \frac{V}{h} = \mu \frac{\omega r}{h}$$

The shear force acting on differential area $dA$ on the surface, the torque it generates, and the shaft power associated with it are expressed as

$$dF = \tau_w dA = \mu \frac{\omega r}{h} dA \qquad d\mathrm{T} = r dF = \mu \frac{\omega r^2}{h} dA$$

$$\mathrm{T} = \frac{\mu\omega}{h}\int_A r^2 dA \qquad \dot{W}_{\text{sh}} = \omega \mathrm{T} = \frac{\mu\omega^2}{h}\int_A r^2 dA$$



**Top surface**: For the top surface, $dA = 2\pi r dr$. Substituting and integrating,

$$\dot{W}_{\text{sh, top}} = \frac{\mu\omega^2}{h}\int_{r=0}^{D/2} r^2 (2\pi r)dr = \frac{2\pi\mu\omega^2}{h}\int_{r=0}^{D/2} r^3 dr = \frac{2\pi\mu\omega^2}{h}\left.\frac{r^4}{4}\right|_{r=0}^{D/2} = \frac{\pi\mu\omega^2 D^4}{32h}$$

**Bottom surface**: A relation for the bottom surface is obtained by replacing $D$ by $d$, $\dot{W}_{\text{sh, bottom}} = \dfrac{\pi\mu\omega^2 d^4}{32h}$

**Side surface**: The differential area for the side surface can be expressed as $dA = 2\pi r dz$. From geometric considerations, the variation of radius with axial distance is expressed as $r = \dfrac{d}{2} + \dfrac{D-d}{2L} z$.

Differentiating gives $dr = \dfrac{D-d}{2L} dz$ or $dz = \dfrac{2L}{D-d} dr$. Therefore, $dA = 2\pi dz = \dfrac{4\pi L}{D-d} r dr$. Substituting and integrating,

$$\dot{W}_{\text{sh, top}} = \frac{\mu\omega^2}{h}\int_{r=0}^{D/2} r^2 \frac{4\pi L}{D-d} r dr = \frac{4\pi\mu\omega^2 L}{h(D-d)}\int_{r=d/2}^{D/2} r^3 dr = \frac{4\pi\mu\omega^2 L}{h(D-d)}\left.\frac{r^4}{4}\right|_{r=d/2}^{D/2} = \frac{\pi\mu\omega^2 L(D^2-d^2)}{16h(D-d)}$$

Then the total power required becomes

$$\dot{W}_{\text{sh, total}} = \dot{W}_{\text{sh, top}} + \dot{W}_{\text{sh, bottom}} + \dot{W}_{\text{sh, side}} = \frac{\pi\mu\omega^2 D^4}{32h}\left[1 + (d/D)^4 + \frac{2L[1-(d/D)^4)]}{D-d}\right],$$

where $d/D$ = 4/12 = 1/3. Substituting,

$$\dot{W}_{\text{sh, total}} = \frac{\pi(0.1\ \text{N}\cdot\text{s/m}^2)(200\ \text{/s})^2(0.12\ \text{m})^4}{32(0.0012\ \text{m})}\left[1 + (1/3)^4 + \frac{2(0.12\ \text{m})[1-(1/3)^4)]}{(0.12-0.04)\ \text{m}}\right]\left(\frac{1\ \text{W}}{1\ \text{Nm/s}}\right) = \mathbf{270\ W}$$

Noting that power is proportional to viscosity, the power required at 80°C is

$$\dot{W}_{\text{sh, total, 80°C}} = \frac{\mu_{80°C}}{\mu_{20°C}}\dot{W}_{\text{sh, total, 20°C}} = \frac{0.0078\ \text{N}\cdot\text{s/m}^2}{0.1\ \text{N}\cdot\text{s/m}^2}(270\ \text{W}) = 21.1\ \text{W}$$

Therefore, the reduction in the requires power input at 80°C is $\text{Reduction} = \dot{W}_{\text{sh, total, 20°C}} - \dot{W}_{\text{sh, total, 80°C}} = 270 - 21.1 = \mathbf{249\ W}$, which is about 92%.

***Discussion*** Note that the power required to overcome shear forces in a viscous fluid greatly depends on temperature.

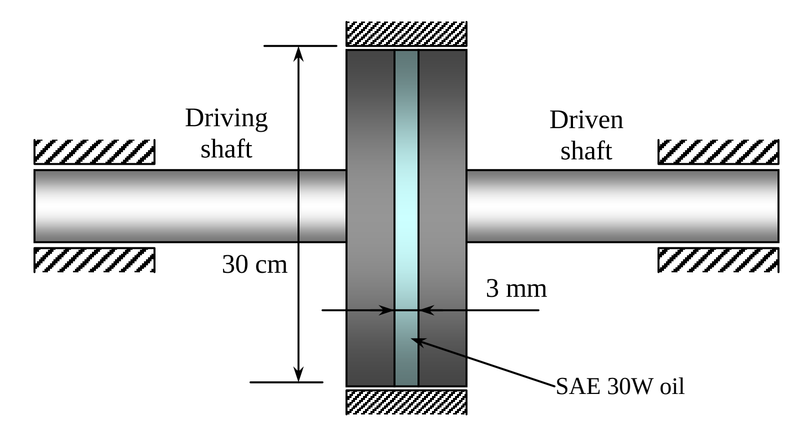


**2-47**

**Solution** A clutch system is used to transmit torque through an oil film between two identical disks. For specified rotational speeds, the transmitted torque is to be determined.

***Assumptions*** **1** The thickness of the oil film is uniform. **2** The rotational speeds of the disks remain constant.

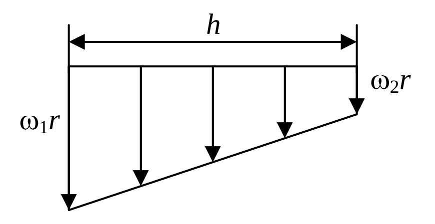
***Properties*** The absolute viscosity of oil is given to be $\mu = 0.38$ N·s/m$^2$.




***Analysis*** The disks are rotting in the same direction at different angular speeds of $\omega_1$ and of $\omega_2$ . Therefore, we can assume one of the disks to be stationary and the other to be rotating at an angular speed of $\omega_1 - \omega_2$. The velocity gradient anywhere in the oil of film thickness *h* is $V\ /h$ where $V = (\omega_1 - \omega_2)r$ is the tangential velocity. Then the wall shear stress anywhere on the surface of the faster disk at a distance *r* from the axis of rotation can be expressed as

$$\tau_w = \mu \frac{du}{dr} = \mu \frac{V}{h} = \mu \frac{(\omega_1 - \omega_2)r}{h}$$

Then the shear force acting on a differential area *dA* on the surface and the torque generation associated with it can be expressed as




$$dF = \tau_w dA = \mu \frac{(\omega_1 - \omega_2)r}{h}(2\pi r)dr$$

$$d\text{T} = rdF = \mu \frac{(\omega_1 - \omega_2)r^2}{h}(2\pi r)dr = \frac{2\pi\mu(\omega_1 - \omega_2)}{h} r^3 dr$$

Integrating,

$$\text{T} = \frac{2\pi\mu(\omega_1 - \omega_2)}{h} \int_{r=0}^{D/2} r^3 dr = \frac{2\pi\mu(\omega_1 - \omega_2)}{h} \left. \frac{r^4}{4} \right|_{r=0}^{D/2} = \frac{\pi\mu(\omega_1 - \omega_2)D^4}{32h}$$

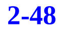
Noting that $\omega = 2\pi\ \dot{n}$, the relative angular speed is

$$\omega_1 - \omega_2 = 2\pi\left(\dot{n}_1 - \dot{n}_2\right) = \left(2\pi \text{ rad/rev}\right)\left[\left(1450 - 1398\right) \text{ rev/min}\right]\left(\frac{1 \text{ min}}{60 \text{ s}}\right) = 5.445 \text{ rad/s},$$

Substituting, the torque transmitted is determined to be

$$\text{T} = \frac{\pi(0.38 \text{ N}\cdot\text{s/m}^2)(5.445\text{ /s})(0.30\text{ m})^4}{32(0.003\text{ m})} = \mathbf{0.55\ N\cdot m}$$

***Discussion*** Note that the torque transmitted is proportional to the fourth power of disk diameter, and is inversely proportional to the thickness of the oil film.

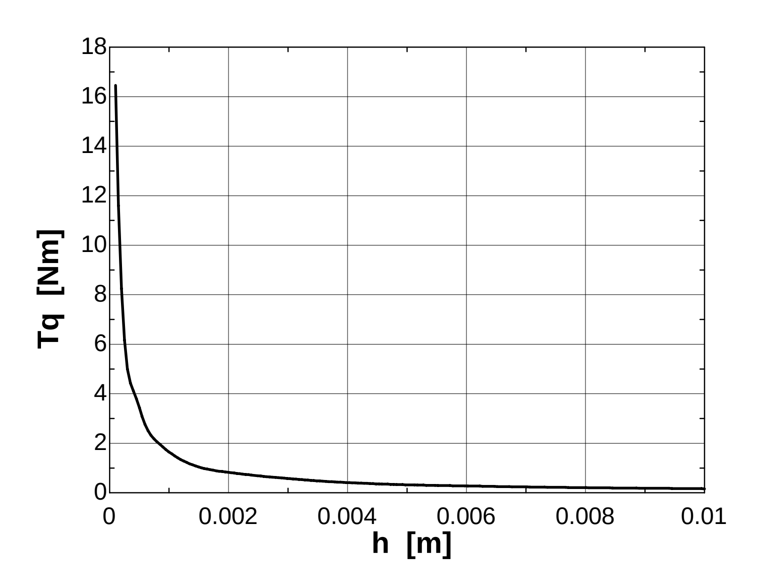

**2-48** 

**Solution** We are to investigate the effect of oil film thickness on the transmitted torque.

***Analysis*** The previous problem is reconsidered. Using EES software, the effect of oil film thickness on the torque transmitted is investigated. Film thickness varied from 0.1 mm to 10 mm, and the results are tabulated and plotted. The relation used is $T = \dfrac{\pi\mu(\omega_1 - \omega_2)D^4}{32h}$. The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
mu=0.38
n1=1450 "rpm"
w1=2*pi*n1/60 "rad/s"
n2=1398 "rpm"
w2=2*pi*n2/60 "rad/s"
D=0.3 "m"
Tq=pi*mu*(w1-w2)*(D^4)/(32*h)
```

| Film thickness *h*, mm | Torque transmitted T, Nm |
|---|---|
| 0.1 | 16.46 |
| 0.2 | 8.23 |
| 0.4 | 4.11 |
| 0.6 | 2.74 |
| 0.8 | 2.06 |
| 1 | 1.65 |
| 2 | 0.82 |
| 4 | 0.41 |
| 6 | 0.27 |
| 8 | 0.21 |
| 10 | 0.16 |




***Conclusion*** Torque transmitted is inversely proportional to oil film thickness, and the film thickness should be as small as possible to maximize the transmitted torque.

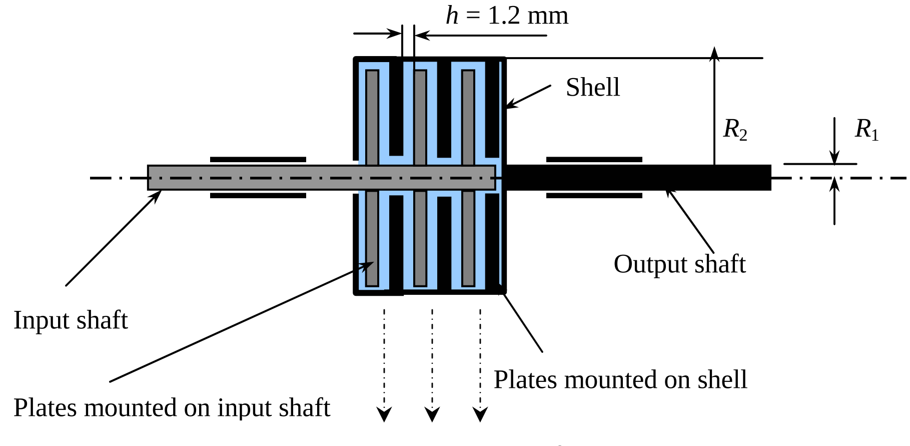
***Discussion*** To obtain the solution in EES, we set up a parametric table, specify *h*, and let EES calculate T for each value of *h*.

**2-49**

**Solution** A multi-disk Electro-rheological "ER" clutch is considered. The ER fluid has a shear stress that is expressed as $\tau = \tau_y + \mu(du/dy)$. A relationship for the torque transmitted by the clutch is to be obtained, and the numerical value of the torque is to be calculated.

***Assumptions*** **1** The thickness of the oil layer between the disks is constant. **2** The Bingham plastic model for shear stress expressed as $\tau = \tau_y + \mu(du/dy)$ is valid.

***Properties*** The constants in shear stress relation are given to be $\mu$ = 0.1 Pa·s and $\tau_y$ = 2.5 kPa.




***Analysis*** (*a*) The velocity gradient anywhere in the oil of film thickness $h$ is $V/h$ where $V = \omega r$ is the tangential velocity relative to plates mounted on the shell. Then the wall shear stress anywhere on the surface of a plate mounted on the input shaft at a distance $r$ from the axis of rotation is expressed as

$$\tau_w = \tau_y + \mu\frac{du}{dr} = \tau_y + \mu\frac{V}{h} = \tau_y + \mu\frac{\omega r}{h}$$

Then the shear force acting on a differential area $dA$ on the surface of a disk and the torque generation associated with it are expressed as

$$dF = \tau_w dA = \left(\tau_y + \mu\frac{\omega r}{h}\right)(2\pi r)dr$$

$$d\mathrm{T} = r dF = r\left(\tau_y + \mu\frac{\omega r}{h}\right)(2\pi r)dr = 2\pi\left(\tau_y r^2 + \mu\frac{\omega r^3}{h}\right)dr$$

Integrating,

$$\mathrm{T} = 2\pi\int_{r=R_1}^{R_2}\left(\tau_y r^2 + \mu\frac{\omega r^3}{h}\right)dr = 2\pi\left[\tau_y\frac{r^3}{3} + \frac{\mu\omega r^4}{4h}\right]_{r=R_1}^{R_2} = 2\pi\left[\frac{\tau_y}{3}(R_2^3 - R_1^3) + \frac{\mu\omega}{4h}(R_2^4 - R_1^4)\right]$$

This is the torque transmitted by one surface of a plate mounted on the input shaft. Then the torque transmitted by both surfaces of $N$ plates attached to input shaft in the clutch becomes

$$\mathrm{T} = 4\pi N\left[\frac{\tau_y}{3}(R_2^3 - R_1^3) + \frac{\mu\omega}{4h}(R_2^4 - R_1^4)\right]$$

(*b*) Noting that $\omega = 2\pi\dot{n} = 2\pi(2400\text{ rev/min}) = 15{,}080\text{ rad/min} = 251.3\text{ rad/s}$ and substituting,

$$\mathrm{T} = (4\pi)(11)\left[\frac{2500\text{ N/m}^2}{3}[(0.20\text{ m})^3 - (0.05\text{ m})^3] + \frac{(0.1\text{ N}\cdot\text{s/m}^2)(251.3\text{ /s})}{4(0.0012\text{ m})}[(0.20\text{ m})^4 - (0.05\text{ m})^4]\right] = \mathbf{2060\ N\cdot m}$$

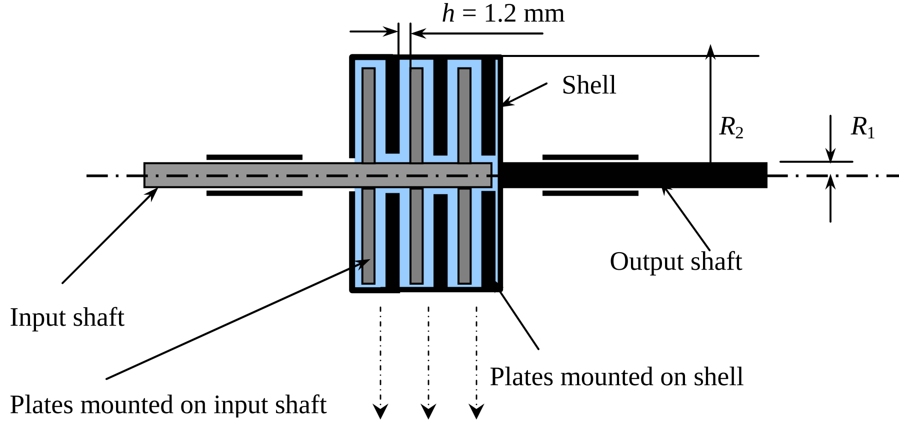
***Discussion*** Can you think of some other potential applications for this kind of fluid?

**2-50**

**Solution** A multi-disk magnetorheological "MR" clutch is considered The MR fluid has a shear stress that is expressed as $\tau = \tau_y + K(du/dy)^m$. A relationship for the torque transmitted by the clutch is to be obtained, and the numerical value of the torque is to be calculated.

***Assumptions*** **1** The thickness of the oil layer between the disks is constant. **2** The Herschel-Bulkley model for shear stress expressed as $\tau = \tau_y + K(du/dy)^m$ is valid.

***Properties*** The constants in shear stress relation are given to be $\tau_y$ = 900 Pa, $K$ = 58 Pa·s$^m$ , and $m$ = 0.82.




***Analysis*** (*a*) The velocity gradient anywhere in the oil of film thickness $h$ is $V/h$ where $V = \omega r$ is the tangential velocity relative to plates mounted on the shell. Then the wall shear stress anywhere on the surface of a plate mounted on the input shaft at a distance $r$ from the axis of rotation is expressed as

$$\tau_w = \tau_y + K\left(\frac{du}{dr}\right)^m = \tau_y + K\left(\frac{V}{h}\right)^m = \tau_y + K\left(\frac{\omega r}{h}\right)^m$$

Then the shear force acting on a differential area $dA$ on the surface of a disk and the torque generation associated with it are expressed as

$$dF = \tau_w dA = \left(\tau_y + K\left(\frac{\omega r}{h}\right)^m\right)(2\pi r)dr \quad \text{and} \quad d\mathrm{T} = rdF = r\left(\tau_y + K\left(\frac{\omega r}{h}\right)^m\right)(2\pi r)dr = 2\pi\left(\tau_y r^2 + K\frac{\omega^m r^{m+2}}{h^m}\right)dr$$

Integrating,

$$\mathrm{T} = 2\pi\int_{R_1}^{R_2}\left(\tau_y r^2 + K\frac{\omega^m r^{m+2}}{h^m}\right)dr = 2\pi\left[\tau_y\frac{r^3}{3} + \frac{K\omega^m r^{m+3}}{(m+3)h^m}\right]_{R_1}^{R_2} = 2\pi\left[\frac{\tau_y}{3}(R_2^3 - R_1^3) + \frac{K\omega^m}{(m+3)h^m}(R_2^{m+3} - R_1^{m+3})\right]$$

This is the torque transmitted by one surface of a plate mounted on the input shaft. Then the torque transmitted by both surfaces of $N$ plates attached to input shaft in the clutch becomes

$$\mathrm{T} = 4\pi N\left[\frac{\tau_y}{3}(R_2^3 - R_1^3) + \frac{K\omega^m}{(m+3)h^m}(R_2^{m+3} - R_1^{m+3})\right]$$

(*b*) Noting that $\omega = 2\pi\dot{n} = 2\pi(2400\text{ rev/min}) = 15{,}080\text{ rad/min} = 251.3\text{ rad/s}$ and substituting,

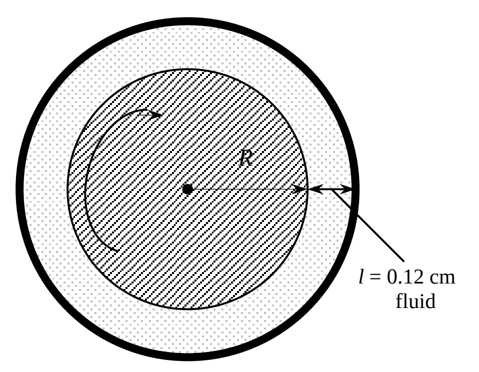
$$\mathrm{T} = (4\pi)(11)\left[\frac{900\text{ N/m}^2}{3}\left[(0.20\text{ m})^3 - (0.05\text{ m})^3\right] + \frac{(58\text{ N}\cdot\text{s}^{0.82}/\text{m}^2)(251.3\text{ /s})^{0.82}}{(0.82+3)(0.0012\text{ m})^{0.82}}\left[(0.20\text{ m})^{3.82} - (0.05\text{ m})^{3.82}\right]\right]$$

$= 103.4$ N $\cong$ **103 kN·m**

***Discussion*** Can you think of some other potential applications for this kind of fluid?

## 2-51

**Solution** The torque and the rpm of a double cylinder viscometer are given. The viscosity of the fluid is to be determined.




***Assumptions*** **1** The inner cylinder is completely submerged in oil. **2** The viscous effects on the two ends of the inner cylinder are negligible. **3** The fluid is Newtonian.

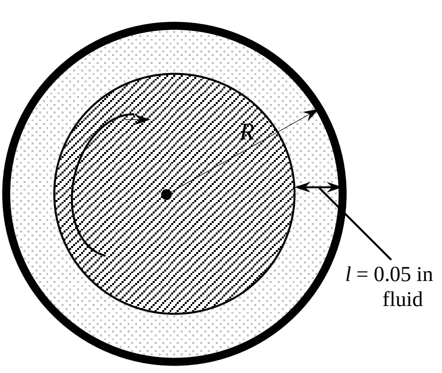
***Analysis*** Substituting the given values, the viscosity of the fluid is determined to be

$$\mu = \frac{\mathbf{T}\ell}{4\pi^2 R^3 \dot{n} L} = \frac{(0.8\ \text{N}\cdot\text{m})(0.0012\ \text{m})}{4\pi^2 (0.075\ \text{m})^3 (200/60\ \text{s}^{-1})(0.75\ \text{m})} = \mathbf{0.0231\ N\cdot s/m^2}$$

***Discussion*** This is the viscosity value at the temperature that existed during the experiment. Viscosity is a strong function of temperature, and the values can be significantly different at different temperatures.

## 2-52E

**Solution** The torque and the rpm of a double cylinder viscometer are given. The viscosity of the fluid is to be determined.




***Assumptions*** **1** The inner cylinder is completely submerged in the fluid. **2** The viscous effects on the two ends of the inner cylinder are negligible. **3** The fluid is Newtonian.

***Analysis*** Substituting the given values, the viscosity of the fluid is determined to be

$$\mu = \frac{\mathbf{T}\ell}{4\pi^2 R^3 \dot{n} L} = \frac{(1.2\ \text{lbf}\cdot\text{ft})(0.05/12\ \text{ft})}{4\pi^2 (5.6/12\ \text{ft})^3 (250/60\ \text{s}^{-1})(3\ \text{ft})} = \mathbf{9.97\times 10^{-5}\ lbf\cdot s/ft^2}$$

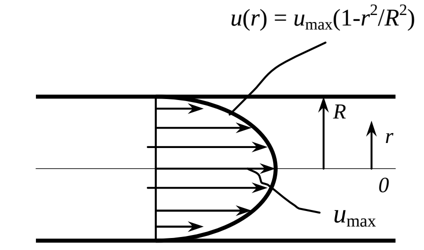
***Discussion*** This is the viscosity value at temperature that existed during the experiment. Viscosity is a strong function of temperature, and the values can be significantly different at different temperatures.

## 2-53

**Solution** The velocity profile for laminar one-dimensional flow through a circular pipe is given. A relation for friction drag force exerted on the pipe and its numerical value for water are to be determined.

***Assumptions*** **1** The flow through the circular pipe is one-dimensional. **2** The fluid is Newtonian.

***Properties*** The viscosity of water at 20°C is given to be 0.0010 kg/m·s.




***Analysis*** The velocity profile is given by $u(r) = u_{max}\left(1 - \frac{r^2}{R^2}\right)$

where $R$ is the radius of the pipe, $r$ is the radial distance from the center of the pipe, and $u_{max}$ is the maximum flow velocity, which occurs at the center, $r$ = 0. The shear stress at the pipe surface is expressed as

$$\tau_w = -\mu \frac{du}{dr}\bigg|_{r=R} = -\mu u_{max} \frac{d}{dr}\left(1 - \frac{r^2}{R^2}\right)_{r=R} = -\mu u_{max} \frac{-2r}{R^2}\bigg|_{r=R} = \frac{2\mu u_{max}}{R}$$

Note that the quantity $du/dr$ is negative in pipe flow, and the negative sign is added to the $\tau_w$ relation for pipes to make shear stress in the positive (flow) direction a positive quantity. (Or, $du/dr = -du/dy$ since $y = R - r$). Then the friction drag force exerted by the fluid on the inner surface of the pipe becomes

$$F_D = \tau_w A_s = \frac{2\mu u_{max}}{R}(2\pi RL) = 4\pi\mu L u_{max}$$

Substituting we get $F_D = 4\pi\mu L u_{max} = 4\pi(0.0010 \text{ kg/m}\cdot\text{s})(15 \text{ m})(3 \text{ m/s})\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = \mathbf{0.565\ N}$

***Discussion*** In the entrance region and during turbulent flow, the velocity gradient is greater near the wall, and thus the drag force in such cases will be greater.

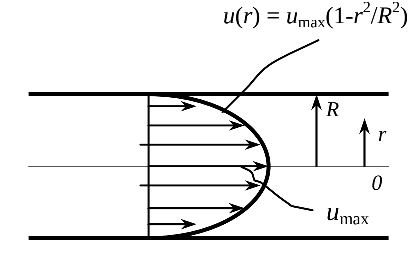
---

## 2-54

**Solution** The velocity profile for laminar one-dimensional flow through a circular pipe is given. A relation for friction drag force exerted on the pipe and its numerical value for water are to be determined.

***Assumptions*** **1** The flow through the circular pipe is one-dimensional. **2** The fluid is Newtonian.

***Properties*** The viscosity of water at 20°C is given to be 0.0010 kg/m·s.




***Analysis*** The velocity profile is given by $u(r) = u_{max}\left(1 - \frac{r^2}{R^2}\right)$

where $R$ is the radius of the pipe, $r$ is the radial distance from the center of the pipe, and $u_{max}$ is the maximum flow velocity, which occurs at the center, $r$ = 0. The shear stress at the pipe surface can be expressed as

$$\tau_w = -\mu \frac{du}{dr}\bigg|_{r=R} = -\mu u_{max} \frac{d}{dr}\left(1 - \frac{r^2}{R^2}\right)_{r=R} = -\mu u_{max} \frac{-2r}{R^2}\bigg|_{r=R} = \frac{2\mu u_{max}}{R}$$

Note that the quantity $du/dr$ is negative in pipe flow, and the negative sign is added to the $\tau_w$ relation for pipes to make shear stress in the positive (flow) direction a positive quantity. (Or, $du/dr = -du/dy$ since $y = R - r$). Then the friction drag force exerted by the fluid on the inner surface of the pipe becomes

$$F_D = \tau_w A_s = \frac{2\mu u_{max}}{R}(2\pi RL) = 4\pi\mu L u_{max}$$

Substituting, we get $F_D = 4\pi\mu L u_{max} = 4\pi(0.0010 \text{ kg/m}\cdot\text{s})(15 \text{ m})(5 \text{ m/s})\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = \mathbf{0.942\ N}$

***Discussion*** In the entrance region and during turbulent flow, the velocity gradient is greater near the wall, and thus the drag force in such cases will be larger.

---

## Surface Tension and Capillary Effect

**2-55C**

**Solution** We are to define and discuss surface tension.

***Analysis*** The **magnitude of the pulling force at the surface of a liquid per unit length** is called *surface tension* $\sigma_s$. It is caused by the attractive forces between the molecules. The surface tension is also surface energy (per unit area) since it represents the stretching work that needs to be done to increase the surface area of the liquid by a unit amount.

***Discussion*** Surface tension is the cause of some very interesting phenomena such as capillary rise and insects that can walk on water.

**2-56C**

**Solution** We are to analyze the pressure difference between inside and outside of a soap bubble.

***Analysis*** The **pressure inside a soap bubble is greater than the pressure outside**, as evidenced by the stretch of the soap film.

***Discussion*** You can make an analogy between the soap film and the skin of a balloon.

**2-57C**

**Solution** We are to define and discuss the capillary effect.

***Analysis*** The *capillary effect* is the **rise or fall of a liquid in a small-diameter tube inserted into the liquid**. It is caused by the net effect of the *cohesive forces* (the forces between like molecules, like water) and *adhesive forces* (the forces between unlike molecules, like water and glass). **The capillary effect is proportional to the cosine of the *contact angle***, which is the angle that the tangent to the liquid surface makes with the solid surface at the point of contact.

***Discussion*** The contact angle determines whether the *meniscus* at the top of the column is concave or convex.

**2-58C**

**Solution** We are to determine whether the level of liquid in a tube will rise or fall due to the capillary effect.

***Analysis*** The **liquid level in the tube will drop** since the contact angle is greater than 90°, and cos(110°) < 0.

***Discussion*** This liquid must be a non-wetting liquid when in contact with the tube material. Mercury is an example of a non-wetting liquid with a contact angle (with glass) that is greater than $90^o$.

**2-59C**

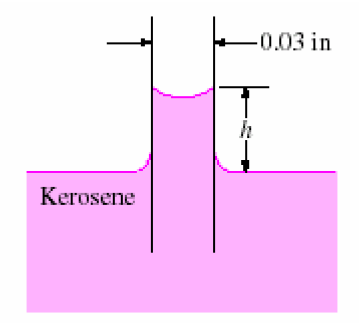
**Solution** We are to compare the capillary rise in small and large diameter tubes.

***Analysis*** The capillary rise is inversely proportional to the diameter of the tube, and thus **capillary rise is greater in the smaller-diameter tube**.

***Discussion*** Note however, that if the tube diameter is large enough, there is no capillary rise (or fall) at all. Rather, the upward (or downward) rise of the liquid occurs only near the tube walls; the elevation of the middle portion of the liquid in the tube does not change for large diameter tubes.

## 2-60E

**Solution** A slender glass tube is inserted into kerosene. The capillary rise of kerosene in the tube is to be determined.




***Assumptions*** **1** There are no impurities in the kerosene, and no contamination on the surfaces of the glass tube. **2** The kerosene is open to the atmospheric air.

***Properties*** The surface tension of kerosene-glass at 68°F (20°C) is $\sigma_s$ = 0.028×0.06852 = 0.00192 lbf/ft. The density of kerosene at 68°F is $\rho$ = 51.2 lbm/ft$^3$. The contact angle of kerosene with the glass surface is given to be 26°.

***Analysis*** Substituting the numerical values, the capillary rise is determined to be



$$h = \frac{2\sigma_s \cos\phi}{\rho g R} = \frac{2(0.00192 \text{ lbf/ft})(\cos 26°)}{(51.2 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)(0.015/12 \text{ ft})}\left(\frac{32.2 \text{ lbm}\cdot\text{ft/s}^2}{1 \text{ lbf}}\right)$$

$$= 0.0539 \text{ ft} = \mathbf{0.650 \text{ in}}$$

***Discussion*** The capillary rise in this case more than half of an inch, and thus it is clearly noticeable.

## 2-61

**Solution** A glass tube is inserted into a liquid, and the capillary rise is measured. The surface tension of the liquid is to be determined.

***Assumptions*** **1** There are no impurities in the liquid, and no contamination on the surfaces of the glass tube. **2** The liquid is open to the atmospheric air.

***Properties*** The density of the liquid is given to be 960 kg/m$^3$. The contact angle is given to be 15°.

***Analysis*** Substituting the numerical values, the surface tension is determined from the capillary rise relation to be

$$\sigma_s = \frac{\rho g R h}{2\cos\phi} = \frac{(960 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(0.0019/2 \text{ m})(0.005 \text{ m})}{2(\cos 15°)}\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = \mathbf{0.0232 \text{ N/m}}$$

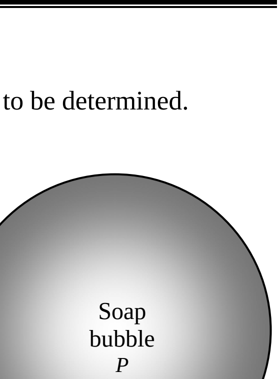
***Discussion*** Since surface tension depends on temperature, the value determined is valid at the liquid's temperature.

## 2-62

**Solution** The diameter of a soap bubble is given. The gage pressure inside the bubble is to be determined.

***Assumptions*** The soap bubble is in atmospheric air.

***Properties*** The surface tension of soap water at 20°C is $\sigma_s$ = 0.025 N/m.




***Analysis*** The pressure difference between the inside and the outside of a bubble is given by

$$\Delta P_{\text{bubble}} = P_i - P_0 = \frac{4\sigma_s}{R}$$

In the open atmosphere $P_0$ = $P_{\text{atm}}$, and thus $\Delta P_{\text{bubble}}$ is equivalent to the gage pressure. Substituting,

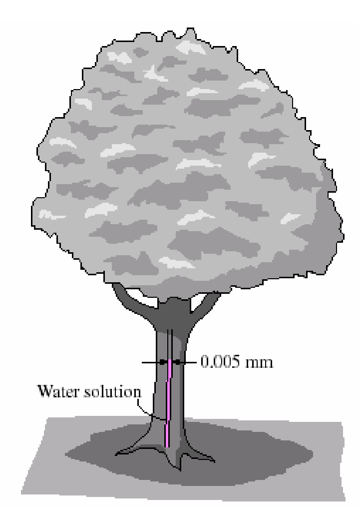
$$P_{i,gage} = \Delta P_{\text{bubble}} = \frac{4(0.025 \text{ N/m})}{0.002/2 \text{ m}} = 100 \text{ N/m}^2 = 100 \text{ Pa}$$

$$P_{i,gage} = \Delta P_{\text{bubble}} = \frac{4(0.025 \text{ N/m})}{0.05/2 \text{ m}} = 4 \text{ N/m}^2 = 4 \text{ Pa}$$

***Discussion*** Note that the gage pressure in a soap bubble is inversely proportional to the radius. Therefore, the excess pressure is larger in smaller bubbles.

## 2-63

**Solution** Nutrients dissolved in water are carried to upper parts of plants. The height to which the water solution rises in a tree as a result of the capillary effect is to be determined.




***Assumptions*** **1** The solution can be treated as water with a contact angle of 15°. **2** The diameter of the tube is constant. **3** The temperature of the water solution is 20°C.

***Properties*** The surface tension of water at 20°C is $\sigma_s$ = 0.073 N/m. The density of water solution can be taken to be 1000 kg/m$^3$. The contact angle is given to be 15°.

***Analysis*** Substituting the numerical values, the capillary rise is determined to be

$$h = \frac{2\sigma_s \cos\phi}{\rho g R} = \frac{2(0.073\,\text{N/m})(\cos 15°)}{(1000\,\text{kg/m}^3)(9.81\,\text{m/s}^2)(2.5\times10^{-6}\,\text{m})}\left(\frac{1\,\text{kg}\cdot\text{m/s}^2}{1\,\text{N}}\right) = \mathbf{5.75\ m}$$

***Discussion*** Other effects such as the chemical potential difference also cause the fluid to rise in trees.

---

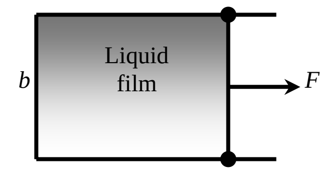
## 2-64

**Solution** The force acting on the movable wire of a liquid film suspended on a U-shaped wire frame is measured. The surface tension of the liquid in the air is to be determined.

***Assumptions*** **1** There are no impurities in the liquid, and no contamination on the surfaces of the wire frame. **2** The liquid is open to the atmospheric air.

***Analysis*** Substituting the numerical values, the surface tension is determined from the surface tension force relation to be

$$\sigma_s = \frac{F}{2b} = \frac{0.012\ \text{N}}{2(0.08\ \text{m})} = \mathbf{0.075\ N/m}$$

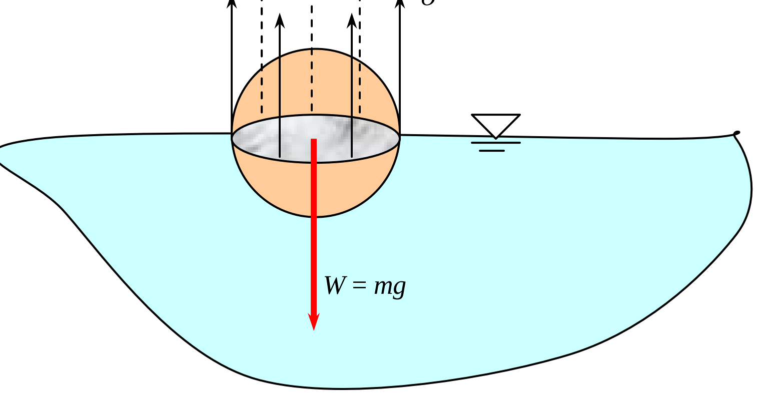



***Discussion*** The surface tension depends on temperature. Therefore, the value determined is valid at the temperature of the liquid.

---

**2-65**

**Solution** A steel ball floats on water due to the surface tension effect. The maximum diameter of the ball is to be determined, and the calculations are to be repeated for aluminum.




***Assumptions*** **1** The water is pure, and its temperature is constant. **2** The ball is dropped on water slowly so that the inertial effects are negligible. **3** The contact angle is taken to be 0° for maximum diameter.

***Properties*** The surface tension of water at 20°C is $\sigma_s$ = 0.073 N/m. The contact angle is taken to be 0°. The densities of steel and aluminum are given to be $\rho_{steel}$ = 7800 kg/m³ and $\rho_{Al}$ = 2700 kg/m³.

***Analysis*** The surface tension force and the weight of the ball can be expressed as

$$F_s = \pi D\sigma_s \quad \text{and} \quad W = mg = \rho g V = \rho g \pi D^3/6$$

When the ball floats, the net force acting on the ball in the vertical direction is zero. Therefore, setting $F_s = W$ and solving for diameter $D$ gives $D = \sqrt{\dfrac{6\sigma_s}{\rho g}}$. Substititing the known quantities, the maximum diameters for the steel and aluminum balls become

$$D_{steel} = \sqrt{\frac{6\sigma_s}{\rho g}} = \sqrt{\frac{6(0.073\ \text{N/m})}{(7800\ \text{kg/m}^3)(9.81\ \text{m/s}^2)}\left(\frac{1\ \text{kg}\cdot\text{m/s}^2}{1\ \text{N}}\right)} = 2.4\times10^{-3}\ \text{m} = \mathbf{2.4\ mm}$$

$$D_{Al} = \sqrt{\frac{6\sigma_s}{\rho g}} = \sqrt{\frac{6(0.073\ \text{N/m})}{(2700\ \text{kg/m}^3)(9.81\ \text{m/s}^2)}\left(\frac{1\ \text{kg}\cdot\text{m/s}^2}{1\ \text{N}}\right)} = 4.1\times10^{-3}\ \text{m} = \mathbf{4.1 mm}$$

***Discussion*** Note that the ball diameter is inversely proportional to the square root of density, and thus for a given material, the smaller balls are more likely to float.

---

## Review Problems

---

**2-66**

**Solution** The pressure in an automobile tire increases during a trip while its volume remains constant. The percent increase in the absolute temperature of the air in the tire is to be determined.

***Assumptions*** **1** The volume of the tire remains constant. **2** Air is an ideal gas.

***Analysis*** Noting that air is an ideal gas and the volume is constant, the ratio of absolute temperatures after and before the trip are

$$\frac{P_1V_1}{T_1} = \frac{P_2V_2}{T_2} \quad \rightarrow \frac{T_2}{T_1} = \frac{P_2}{P_1} = \frac{310\ \text{kPa}}{290\ \text{kPa}} = 1.069$$

Therefore, the absolute temperature of air in the tire will increase by **6.9%** during this trip.

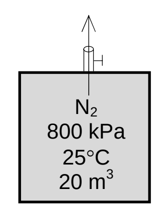
***Discussion*** This may not seem like a large temperature increase, but if the tire is originally at 20°C (293.15 K), the temperature increases to 1.069(293.15 K) = 313.38 K or about 40.2°C.

## 2-67

**Solution** A large tank contains nitrogen at a specified temperature and pressure. Now some nitrogen is allowed to escape, and the temperature and pressure of nitrogen drop to new values. The amount of nitrogen that has escaped is to be determined.

***Assumptions*** The tank is insulated so that no heat is transferred.

***Analysis*** Treating $N_2$ as an ideal gas, the initial and the final masses in the tank are determined to be




$$m_1 = \frac{P_1 V}{RT_1} = \frac{(800\text{kPa})(20\text{m}^3)}{(0.2968\text{kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(298\text{K})} = 180.9\text{kg}$$

$$m_2 = \frac{P_2 V}{RT_2} = \frac{(600\text{kPa})(20\text{m}^3)}{(0.2968\text{kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(293\text{K})} = 138.0\text{kg}$$

Thus the amount of $N_2$ that escaped is $\Delta m = m_1 - m_2 = 180.9 - 138.0 = \mathbf{42.9\ kg}$

***Discussion*** Gas expansion generally causes the temperature to drop. This principle is used in some types of refrigeration.

---

## 2-68

**Solution** Suspended solid particles in water are considered. A relation is to be developed for the specific gravity of the suspension in terms of the mass fraction $C_{s,\text{ mass}}$ and volume fraction $C_{s,\text{ vol}}$ of the particles.

***Assumptions*** **1** The solid particles are distributed uniformly in water so that the solution is homogeneous. **2** The effect of dissimilar molecules on each other is negligible.

***Analysis*** Consider solid particles of mass $m_s$ and volume $V_s$ dissolved in a fluid of mass $m_f$ and volume $V_m$. The total volume of the suspension (or mixture) is

$$V_m = V_s + V_f$$

Dividing by $V_m$ and using the definition $C_{s,\text{ vol}} = V_s / V_m$ give

$$1 = C_{s,vol} + \frac{V_f}{V_m} \quad \rightarrow \quad \frac{V_f}{V_m} = 1 - C_{s,vol} \qquad (1)$$

The total mass of the suspension (or mixture) is

$$m_m = m_s + m_f$$

Dividing by $m_m$ and using the definition $C_{s,\text{ mass}} = m_s / m_m$ give

$$1 = C_{s,mass} + \frac{m_f}{m_m} = C_{s,mass} + \frac{\rho_f V_f}{\rho_m V_m} \quad \rightarrow \quad \frac{\rho_f}{\rho_m} = (1 - C_{s,mass})\frac{V_m}{V_f} \qquad (2)$$

Combining equations 1 and 2 gives

$$\frac{\rho_f}{\rho_m} = \frac{1 - C_{s,mass}}{1 - C_{s,vol}}$$

When the fluid is water, the ratio $\rho_f / \rho_m$ is the inverse of the definition of specific gravity. Therefore, the desired relation for the specific gravity of the mixture is

$$\boxed{\text{SG}_m = \frac{\rho_m}{\rho_f} = \frac{1 - C_{s,vol}}{1 - C_{s,mass}}}$$

which is the desired result.

***Discussion*** As a quick check, if there were no particles at all, $\text{SG}_m = 0$, as expected.

---

## 2-69

**Solution** The specific gravities of solid particles and carrier fluids of a slurry are given. The relation for the specific gravity of the slurry is to be obtained in terms of the mass fraction $C_{s,\text{ mass}}$ and the specific gravity $SG_s$ of solid particles.

***Assumptions*** **1** The solid particles are distributed uniformly in water so that the solution is homogeneous. **2** The effect of dissimilar molecules on each other is negligible.

***Analysis*** Consider solid particles of mass $m_s$ and volume $V_s$ dissolved in a fluid of mass $m_f$ and volume $V_m$. The total volume of the suspension (or mixture) is $V_m = V_s + V_f$.

Dividing by $V_m$ gives

$$1 = \frac{V_s}{V_m} + \frac{V_f}{V_m} \quad \rightarrow \quad \frac{V_f}{V_m} = 1 - \frac{V_s}{V_m} = 1 - \frac{m_s/\rho_s}{m_m/\rho_m} = 1 - \frac{m_s}{m_m}\frac{\rho_m}{\rho_s} = 1 - C_{s,\text{mass}}\frac{SG_m}{SG_s} \qquad (1)$$

since ratio of densities is equal two the ratio of specific gravities, and $m_s / m_m = C_{\text{s, mass}}$. The total mass of the suspension (or mixture) is $m_m = m_s + m_f$. Dividing by $m_m$ and using the definition $C_{\text{s, mass}} = m_s / m_m$ give

$$1 = C_{s,\text{mass}} + \frac{m_f}{m_m} = C_{s,\text{mass}} + \frac{\rho_f V_f}{\rho_m V_m} \quad \rightarrow \quad \frac{\rho_m}{\rho_f} = \frac{V_f}{(1 - C_{s,mass})V_m} \qquad (2)$$

Taking the fluid to be water so that $\rho_m / \rho_f = SG_m$ and combining equations 1 and 2 give

$$SG_m = \frac{1 - C_{s,\text{mass}} SG_m / SG_s}{1 - C_{s,\text{mass}}}$$

Solving for $SG_m$ and rearranging gives

$$\boxed{SG_m = \frac{1}{1 + C_{\text{s, mass}}(1/SG_s - 1)}}$$

which is the desired result.

***Discussion*** As a quick check, if there were no particles at all, $SG_m = 0$, as expected.

---

## 2-70E

**Solution** The minimum pressure on the suction side of a water pump is given. The maximum water temperature to avoid the danger of cavitation is to be determined.

***Properties*** The saturation temperature of water at 0.95 psia is 100°F.

***Analysis*** To avoid cavitation at a specified pressure, the fluid temperature everywhere in the flow should remain below the saturation temperature at the given pressure, which is

$$T_{\max} = T_{\text{sat @ 0.95 psia}} = \mathbf{100°F}$$

Therefore, ***T* must remain below 100°F to avoid the possibility of cavitation**.

***Discussion*** Note that saturation temperature increases with pressure, and thus cavitation may occur at higher pressure at locations with higher fluid temperatures.

---

## 2-71

**Solution** Air in a partially filled closed water tank is evacuated. The absolute pressure in the evacuated space is to be determined.

***Properties*** The saturation pressure of water at 60°C is 19.94 kPa.

***Analysis*** When air is completely evacuated, the vacated space is filled with water vapor, and the tank contains a saturated water-vapor mixture at the given pressure. Since we have a two-phase mixture of a pure substance at a specified temperature, the vapor pressure must be the saturation pressure at this temperature. That is,

$$P_v = P_{\text{sat @ 60°C}} = 19.94 \text{ kPa} \cong \mathbf{19.9 \ kPa}$$

***Discussion*** If there is any air left in the container, the vapor pressure will be less. In that case the sum of the component pressures of vapor and air would equal 19.94 kPa.

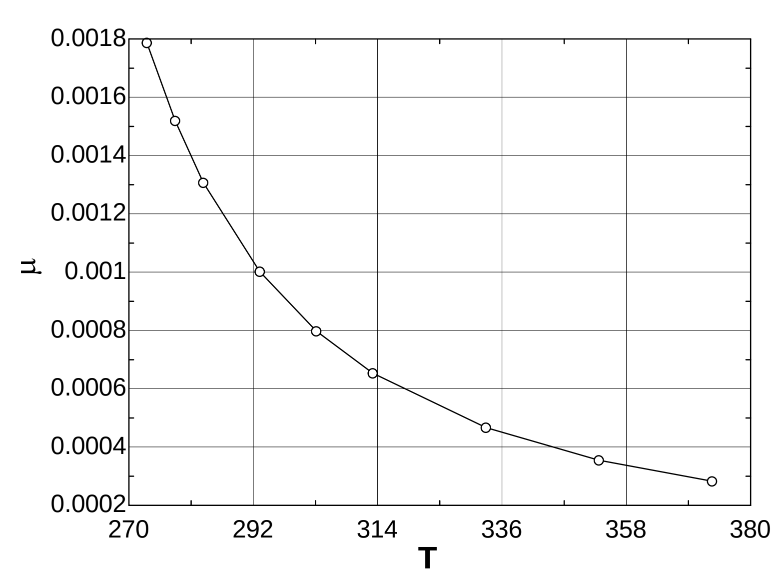
---

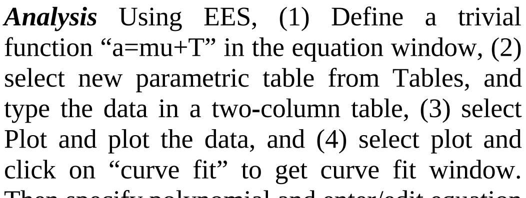
## 2-72 

**Solution** The variation of the dynamic viscosity of water with absolute temperature is given. Using tabular data, a relation is to be obtained for viscosity as a 4th-order polynomial. The result is to be compared to Andrade's equation in the form of $\mu = D \cdot e^{B/T}$.

***Properties*** The viscosity data are given in tabular form as

| $T$ (K) | $\mu$ (Pa·s) |
|---|---|
| 273.15 | $1.787\times10^{-3}$ |
| 278.15 | $1.519\times10^{-3}$ |
| 283.15 | $1.307\times10^{-3}$ |
| 293.15 | $1.002\times10^{-3}$ |
| 303.15 | $7.975\times10^{-4}$ |
| 313.15 | $6.529\times10^{-4}$ |
| 333.15 | $4.665\times10^{-4}$ |
| 353.15 | $3.547\times10^{-4}$ |
| 373.15 | $2.828\times10^{-4}$ |



***Analysis*** Using EES, (1) Define a trivial function "a=mu+T" in the equation window, (2) select new parametric table from Tables, and type the data in a two-column table, (3) select Plot and plot the data, and (4) select plot and click on "curve fit" to get curve fit window. Then specify polynomial and enter/edit equation. The equations and plot are shown here.

$\mu$ = 0.489291758 - 0.00568904387$T$ + 0.0000249152104$T^2$ - 4.86155745$\times10^{-8}T^3$ + 3.56198079$\times10^{-11}T^4$
$\mu$ = 0.000001475*EXP(1926.5/$T$) [used initial guess of a0=1.8$\times10^{-6}$ and a1=1800 in mu=a0*exp(a1/T)]

At $T$ = 323.15 K, the polynomial and exponential curve fits give

*Polynomial*: $\mu$(323.15 K) = 0.0005529 Pa·s (1.1% error, relative to 0.0005468 Pa·s)
*Exponential*: $\mu$(323.15 K) = 0.0005726 Pa·s (4.7% error, relative to 0.0005468 Pa·s)

***Discussion*** This problem can also be solved using an Excel worksheet, with the following results:
Polynomial: **A = 0.4893, B = -0.005689, C = 0.00002492, D = -0.000000048612, and E = 0.00000000003562**
Andrade's equation: $\boxed{\mu = 1.807952E-6 * e^{1864.06/T}}$

---

**2-73**

**Solution** The velocity profile for laminar one-dimensional flow between two parallel plates is given. A relation for friction drag force exerted on the plates per unit area of the plates is to be obtained.

***Assumptions*** **1** The flow between the plates is one-dimensional. **2** The fluid is Newtonian.

$u(y) = 4u_{\max}\left[y/h - (y/h)^2\right]$

***Analysis*** The velocity profile is given by $u(y) = 4u_{\max}\left[y/h - (y/h)^2\right]$

where $h$ is the distance between the two plates, $y$ is the vertical distance from the bottom plate, and $u_{\max}$ is the maximum flow velocity that occurs at midplane. The shear stress at the bottom surface can be expressed as

$$\tau_w = \mu \frac{du}{dy}\bigg|_{y=0} = 4\mu u_{\max} \frac{d}{dy}\left(\frac{y}{h} - \frac{y^2}{h^2}\right)_{y=0} = 4\mu u_{\max}\left(\frac{1}{h} - \frac{2y}{h^2}\right)\bigg|_{y=0} = \frac{4\mu u_{\max}}{h}$$

Because of symmetry, the wall shear stress is identical at both bottom and top plates. Then the friction drag force exerted by the fluid on the inner surface of the plates becomes

$$F_D = 2\tau_w A_{plate} = \frac{8\mu u_{\max}}{h} A_{\text{plate}}$$

Therefore, the friction drag per unit plate area is

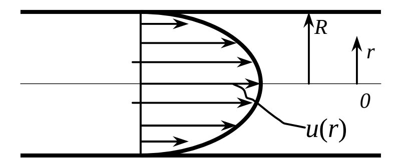
$$\boxed{F_D / A_{\text{plate}} = \frac{8\mu u_{\max}}{h}}$$

***Discussion*** Note that the friction drag force acting on the plates is inversely proportional to the distance between plates.

---

**2-74**

**Solution** The laminar flow of a Bingham plastic fluid in a horizontal pipe of radius $R$ is considered. The shear stress at the pipe wall and the friction drag force acting on a pipe section of length $L$ are to be determined.




***Assumptions*** **1** The fluid is a Bingham plastic with $\tau = \tau_y + \mu(du/dr)$ where $\tau_y$ is the yield stress. **2** The flow through the pipe is one-dimensional.

***Analysis*** The velocity profile is given by $u(r) = \frac{\Delta P}{4\mu L}(r^2 - R^2) + \frac{\tau_y}{\mu}(r - R)$ where $\Delta P/L$ is the pressure drop along the pipe per unit length, $\mu$ is the dynamic viscosity, $r$ is the radial distance from the centerline. Its gradient at the pipe wall ($r = R$) is

$$\frac{du}{dr}\bigg|_{r=R} = \frac{d}{dr}\left(\frac{\Delta P}{4\mu L}(r^2 - R^2) + \frac{\tau_y}{\mu}(r - R)\right)\bigg|_{r=R} = \left(2r\frac{\Delta P}{4\mu L} + \frac{\tau_y}{\mu}\right)_{r=R} = \frac{1}{\mu}\left(\frac{\Delta P}{2L}R + \tau_y\right)$$

Substituing into $\tau = \tau_y + \mu(du/dr)$, the wall shear stress at the pipe surface becomes

$$\tau_w = \tau_y + \mu\frac{du}{dr}\bigg|_{r=R} = \tau_y + \frac{\Delta P}{2L}R + \tau_y = 2\tau_y + \frac{\Delta P}{2L}R$$

Then the friction drag force exerted by the fluid on the inner surface of the pipe becomes

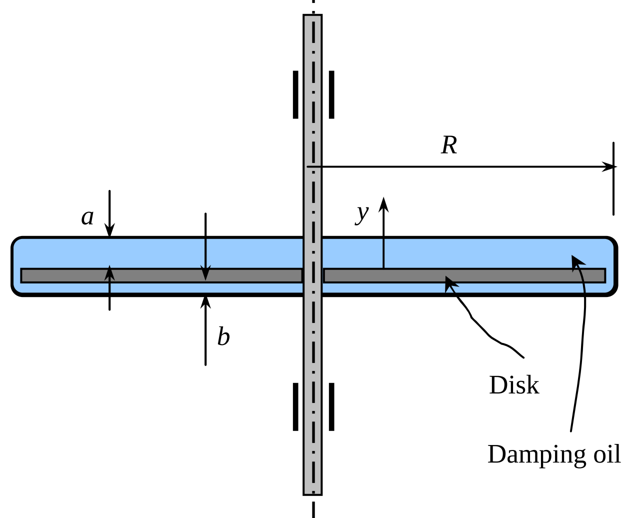
$$\boxed{F_D = \tau_w A_s = \left(2\tau_y + \frac{\Delta P}{2L}R\right)(2\pi RL) = 2\pi RL\left(2\tau_y + \frac{\Delta P}{2L}R\right) = 4\pi RL\,\tau_y + \pi R^2 \Delta P}$$

***Discussion*** Note that the total friction drag is proportional to yield shear stress and the pressure drop.

**2-75**

**Solution** A circular disk immersed in oil is used as a damper, as shown in the figure. It is to be shown that the damping torque is $\mathrm{T}_{\text{damping}} = C\omega$ where $C = 0.5\pi\mu(1/a + 1/b)R^4$.

***Assumptions*** **1** The thickness of the oil layer on each side remains constant. **2** The velocity profiles are linear on both sides of the disk. **3** The tip effects are negligible. **4** The effect of the shaft is negligible.




***Analysis*** The velocity gradient anywhere in the oil of film thickness $a$ is $V/a$ where $V = \omega r$ is the tangential velocity. Then the wall shear stress anywhere on the upper surface of the disk at a distance $r$ from the axis of rotation can be expressed as

$$\tau_w = \mu\frac{du}{dy} = \mu\frac{V}{a} = \mu\frac{\omega r}{a}$$

Then the shear force acting on a differential area $dA$ on the surface and the torque it generates can be expressed as

$$dF = \tau_w dA = \mu\frac{\omega r}{a}dA$$

$$d\mathrm{T} = r dF = \mu\frac{\omega r^2}{a}dA$$


Noting that $dA = 2\pi r dr$ and integrating, the torque on the top surface is determined to be

$$\mathrm{T}_{\text{top}} = \frac{\mu\omega}{a}\int_A r^2 dA = \frac{\mu\omega}{a}\int_{r=0}^{R} r^2(2\pi r)dr = \frac{2\pi\mu\omega}{a}\int_{r=0}^{R} r^3 dr = \frac{2\pi\mu\omega}{a}\left.\frac{r^4}{4}\right|_{r=0}^{R} = \frac{\pi\mu\omega R^4}{2a}$$

The torque on the bottom surface is obtained by replaying $a$ by $b$,

$$\mathrm{T}_{\text{bottom}} = \frac{\pi\mu\omega R^4}{2b}$$

The total torque acting on the disk is the sum of the torques acting on the top and bottom surfaces,

$$\mathrm{T}_{\text{damping, total}} = \mathrm{T}_{\text{bottom}} + \mathrm{T}_{\text{top}} = \frac{\pi\mu\omega R^4}{2}\left(\frac{1}{a} + \frac{1}{b}\right)$$

or,

$$\mathrm{T}_{\text{damping,total}} = C\omega \quad \text{where} \quad C = \frac{\pi\mu R^4}{2}\left(\frac{1}{a} + \frac{1}{b}\right)$$

This completes the proof.

***Discussion*** Note that the damping torque (and thus damping power) is inversely proportional to the thickness of oil films on either side, and it is proportional to the 4th power of the radius of the damper disk.

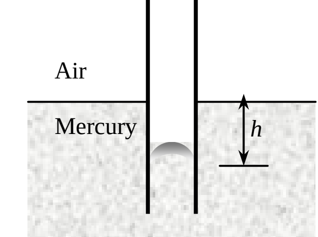
**2-76E**

**Solution** A glass tube is inserted into mercury. The capillary drop of mercury in the tube is to be determined.

***Assumptions*** **1** There are no impurities in mercury, and no contamination on the surfaces of the glass tube. **2** The mercury is open to the atmospheric air.

***Properties*** The surface tension of mercury-glass in atmospheric air at 68°F (20°C) is $\sigma_s$ = 0.440×0.06852 = 0.03015 lbf/ft. The density of mercury is $\rho$ = 847 lbm/ft$^3$ at 77°F, but we can also use this value at 68°F. The contact angle is given to be 140°.

***Analysis*** Substituting the numerical values, the capillary drop is determined to be




$$h = \frac{2\sigma_s \cos\phi}{\rho g R} = \frac{2(0.03015 \text{ lbf/ft})(\cos 140°)}{(847 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)(0.45/12 \text{ ft})}\left(\frac{32.2 \text{ lbm}\cdot\text{ft/s}^2}{1 \text{ lbf}}\right)$$

$$= -0.00145 \text{ ft} = \mathbf{-0.0175 \text{ in}}$$

***Discussion*** The negative sign indicates capillary drop instead of rise. The drop is very small in this case because of the large diameter of the tube.

**2-77**

**Solution** A relation is to be derived for the capillary rise of a liquid between two large parallel plates a distance *t* apart inserted into a liquid vertically. The contact angle is given to be $\phi$.

***Assumptions*** There are no impurities in the liquid, and no contamination on the surfaces of the plates.

***Analysis*** The magnitude of the capillary rise between two large parallel plates can be determined from a force balance on the rectangular liquid column of height *h* and width *w* between the plates. The bottom of the liquid column is at the same level as the free surface of the liquid reservoir, and thus the pressure there must be atmospheric pressure. This will balance the atmospheric pressure acting from the top surface, and thus these two effects will cancel each other. The weight of the liquid column is

$$W = mg = \rho g V = \rho g(w \times t \times h)$$

Equating the vertical component of the surface tension force to the weight gives

$$W = F_{surface} \quad \rightarrow \quad \rho g(w \times t \times h) = 2w\sigma_s \cos\phi$$

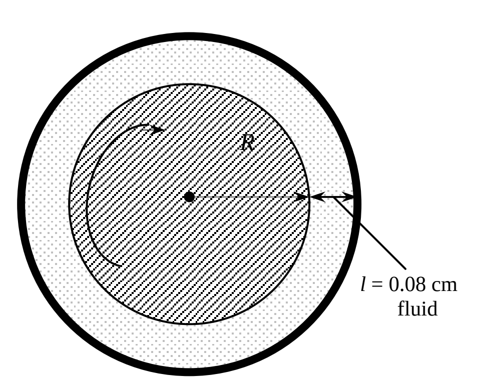
Canceling *w* and solving for *h* gives the capillary rise to be

*Capillary rise*: $$h = \frac{2\sigma_s \cos\phi}{\rho g t}$$

***Discussion*** The relation above is also valid for non-wetting liquids (such as mercury in glass), and gives a capillary drop instead of a capillary rise.

**2-78**

**Solution** A journal bearing is lubricated with oil whose viscosity is known. The torques needed to overcome the bearing friction during start-up and steady operation are to be determined.




***Assumptions*** **1** The gap is uniform, and is completely filled with oil. **2** The end effects on the sides of the bearing are negligible. **3** The fluid is Newtonian.

***Properties*** The viscosity of oil is given to be 0.1 kg/m·s at 20°C, and 0.008 kg/m·s at 80°C.

***Analysis*** The radius of the shaft is $R$ = 0.04 m. Substituting the given values, the torque is determined to be

*At start up at* 20°C:

$$\mathbf{T} = \mu \frac{4\pi^2 R^3 \dot{n} L}{\ell} = (0.1\,\text{kg/m}\cdot\text{s}) \frac{4\pi^2 (0.04\,\text{m})^3 (500/60\,\text{s}^{-1})(0.30\,\text{m})}{0.0008\,\text{m}} = \mathbf{0.79\,N\cdot m}$$

*During steady operation at* 80°C:

$$\mathbf{T} = \mu \frac{4\pi^2 R^3 \dot{n} L}{\ell} = (0.008\,\text{kg/m}\cdot\text{s}) \frac{4\pi^2 (0.04\,\text{m})^3 (500/60\,\text{s}^{-1})(0.30\,\text{m})}{0.0008\,\text{m}} = \mathbf{0.063\,N\cdot m}$$

***Discussion*** Note that the torque needed to overcome friction reduces considerably due to the decrease in the viscosity of oil at higher temperature.

---

## Design and Essay Problems

**2-79 to 2-81**

**Solution** Students' essays and designs should be unique and will differ from each other.

---



# Solutions Manual for
# Fluid Mechanics: Fundamentals and Applications
# by Çengel & Cimbala

---

# CHAPTER 3
# PRESSURE AND FLUID STATICS

---

## Pressure, Manometer, and Barometer

### 3-1C

**Solution** We are to discuss the difference between gage pressure and absolute pressure.

***Analysis*** The **pressure relative to the atmospheric pressure** is called the *gage pressure*, and the **pressure relative to an absolute vacuum** is called *absolute pressure*.

***Discussion*** Most pressure gages (like your bicycle tire gage) read relative to atmospheric pressure, and therefore read the gage pressure.

### 3-2C

**Solution** We are to explain nose bleeding and shortness of breath at high elevation.

***Analysis*** Atmospheric air pressure which is the external pressure exerted on the skin decreases with increasing elevation. Therefore, **the pressure is lower at higher elevations. As a result, the difference between the blood pressure in the veins and the air pressure outside increases**. **This pressure imbalance may cause some thin-walled veins such as the ones in the nose to burst, causing bleeding**. The shortness of breath is caused by the lower air density at higher elevations, and thus lower amount of oxygen per unit volume.

***Discussion*** People who climb high mountains like Mt. Everest suffer other physical problems due to the low pressure.

### 3-3C

**Solution** We are to examine a claim about absolute pressure.

***Analysis*** **No, the absolute pressure in a liquid of constant density does not double when the depth is doubled**. It is the *gage pressure* that doubles when the depth is doubled.

***Discussion*** This is analogous to temperature scales – when performing analysis using something like the ideal gas law, you *must* use absolute temperature (K), not relative temperature ($^{o}$C), or you will run into the same kind of problem.

### 3-4C

**Solution** We are to compare the pressure on the surfaces of a cube.

***Analysis*** Since pressure increases with depth, **the pressure on the bottom face of the cube is higher than that on the top. The pressure varies linearly along the side faces**. However, if the lengths of the sides of the tiny cube suspended in water by a string are very small, the magnitudes of the pressures on all sides of the cube are nearly the same.

***Discussion*** In the limit of an "infinitesimal cube", we have a fluid particle, with pressure $P$ defined at a "point".

### 3-5C

**Solution** We are to define Pascal's law and give an example.

***Analysis*** *Pascal's law* states that **the pressure applied to a confined fluid increases the pressure throughout by the same amount**. This is a consequence of the pressure in a fluid remaining constant in the horizontal direction. An example of Pascal's principle is the operation of the hydraulic car jack.

***Discussion*** The above discussion applies to fluids at rest (hydrostatics). When fluids are in motion, Pascal's principle does not necessarily apply. However, as we shall see in later chapters, the differential equations of incompressible fluid flow contain only pressure *gradients*, and thus an increase in pressure *in the whole system* does *not* affect fluid motion.

**3-6C**

**Solution** We are to compare the volume and mass flow rates of two fans at different elevations.

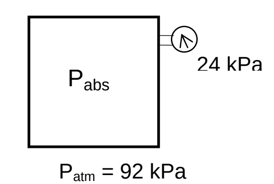
***Analysis*** The density of air at sea level is higher than the density of air on top of a high mountain. Therefore, the volume flow rates of the two fans running at identical speeds will be the same, but the mass flow rate of the fan at sea level will be higher.

***Discussion*** In reality, the fan blades on the high mountain would experience less frictional drag, and hence the fan motor would not have as much resistance – the rotational speed of the fan on the mountain would be slightly higher than that at sea level.

---

**3-7**

**Solution** The pressure in a vacuum chamber is measured by a vacuum gage. The absolute pressure in the chamber is to be determined.




***Analysis*** The absolute pressure in the chamber is determined from

$$P_{abs} = P_{atm} - P_{vac} = 92 - 24 = \mathbf{68\ kPa}$$

***Discussion*** We must remember that "vacuum pressure" is the negative of gage pressure – hence the negative sign.

---

**3-8E**

**Solution** The pressure in a tank is measured with a manometer by measuring the differential height of the manometer fluid. The absolute pressure in the tank is to be determined for two cases: the manometer arm with the (*a*) higher and (*b*) lower fluid level being attached to the tank.

***Assumptions*** The fluid in the manometer is incompressible.

***Properties*** The specific gravity of the fluid is given to be SG = 1.25. The density of water at 32°F is 62.4 lbm/ft$^3$.

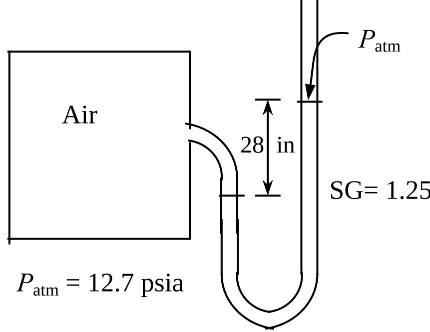
***Analysis*** The density of the fluid is obtained by multiplying its specific gravity by the density of water,

$$\rho = \text{SG} \times \rho_{H_2O} = (1.25)(62.4\ \text{lbm/ft}^3) = 78.0\ \ \text{lbm/ft}^3$$

The pressure difference corresponding to a differential height of 28 in between the two arms of the manometer is

$$\Delta P = \rho g h = (78\,\text{lbm/ft}^3)(32.174\,\text{ft/s}^2)(28/12\,\text{ft})\left(\frac{1\,\text{lbf}}{32.174\,\text{lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1\,\text{ft}^2}{144\,\text{in}^2}\right) = 1.26\,\text{psia}$$

Then the absolute pressures in the tank for the two cases become:




(*a*) The fluid level in the arm attached to the tank is higher (vacuum):

$$P_{abs} = P_{atm} - P_{vac} = 12.7 - 1.26 = 11.44\ \text{psia} \cong \mathbf{11.4\ psia}$$

(*b*) The fluid level in the arm attached to the tank is lower:

$$P_{abs} = P_{gage} + P_{atm} = 12.7 + 1.26 = 13.96\ \text{psia} \cong \mathbf{14.0\ \ psia}$$

***Discussion*** The final results are reported to three significant digits. Note that we can determine whether the pressure in a tank is above or below atmospheric pressure by simply observing the side of the manometer arm with the higher fluid level.

---

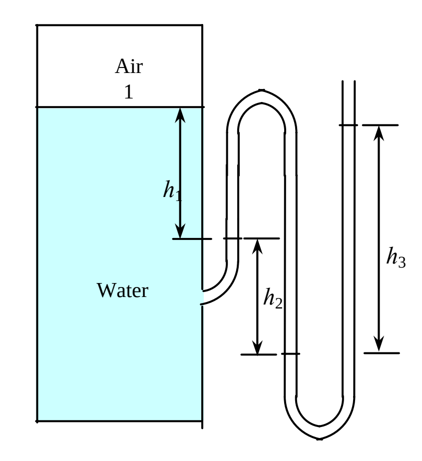
**3-9**

**Solution** The pressure in a pressurized water tank is measured by a multi-fluid manometer. The gage pressure of air in the tank is to be determined.

***Assumptions*** The air pressure in the tank is uniform (i.e., its variation with elevation is negligible due to its low density), and thus we can determine the pressure at the air-water interface.

***Properties*** The densities of mercury, water, and oil are given to be 13,600, 1000, and 850 kg/m$^3$, respectively.

***Analysis*** Starting with the pressure at point 1 at the air-water interface, and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach point 2, and setting the result equal to $P_{\text{atm}}$ since the tube is open to the atmosphere gives




$$P_1 + \rho_{\text{water}} gh_1 + \rho_{\text{oil}} gh_2 - \rho_{\text{mercury}} gh_3 = P_{atm}$$

Solving for $P_{1,}$

$$P_1 = P_{\text{atm}} - \rho_{\text{water}} gh_1 - \rho_{\text{oil}} gh_2 + \rho_{\text{mercury}} gh_3$$

or,

$$P_1 - P_{\text{atm}} = g(\rho_{\text{mercury}} h_3 - \rho_{\text{water}} h_1 - \rho_{\text{oil}} h_2)$$

Noting that $P_{1,\text{gage}} = P_1 - P_{\text{atm}}$ and substituting,

$$P_{1,gage} = (9.81\ \text{m/s}^2)[(13{,}600\ \text{kg/m}^3)(0.46\ \text{m}) - (1000\ \text{kg/m}^3)(0.2\ \text{m})$$

$$-(850\ \text{kg/m}^3)(0.3\ \text{m})]\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1000\ \text{N/m}^2}\right)$$

$$= \mathbf{56.9\ kPa}$$

***Discussion*** Note that jumping horizontally from one tube to the next and realizing that pressure remains the same in the same fluid simplifies the analysis greatly.

---

**3-10**

**Solution** The barometric reading at a location is given in height of mercury column. The atmospheric pressure is to be determined.

***Properties*** The density of mercury is given to be 13,600 kg/m$^3$.

***Analysis*** The atmospheric pressure is determined directly from

$$P_{atm} = \rho gh = \left(13{,}600\ \text{kg/m}^3\right)\left(9.81\ \text{m/s}^2\right)\left(0.750\ \text{m}\right)\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1000\ \text{N/m}^2}\right)$$

$$= 100.1\ \text{kPa} \cong \mathbf{100\ \ kPa}$$

***Discussion*** We round off the final answer to three significant digits. 100 kPa is a fairly typical value of atmospheric pressure on land slightly above sea level.

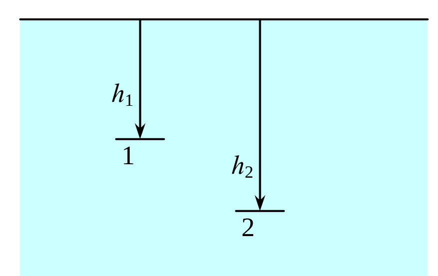
---

**3-11**

**Solution** The gage pressure in a liquid at a certain depth is given. The gage pressure in the same liquid at a different depth is to be determined.

***Assumptions*** The variation of the density of the liquid with depth is negligible.

***Analysis*** The gage pressure at two different depths of a liquid can be expressed as $P_1 = \rho g h_1$ and $P_2 = \rho g h_2$. Taking their ratio,




$$\frac{P_2}{P_1} = \frac{\rho g h_2}{\rho g h_1} = \frac{h_2}{h_1}$$

Solving for $P_2$ and substituting gives

$$P_2 = \frac{h_2}{h_1} P_1 = \frac{12 \text{ m}}{3 \text{ m}} (28 \text{ kPa}) = \mathbf{112 \ kPa}$$

***Discussion*** Note that the gage pressure in a given fluid is proportional to depth.

**3-12**

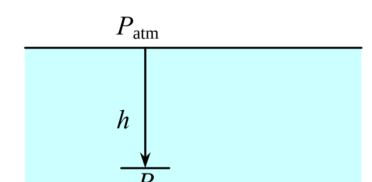
**Solution** The absolute pressure in water at a specified depth is given. The local atmospheric pressure and the absolute pressure at the same depth in a different liquid are to be determined.

***Assumptions*** The liquid and water are incompressible.

***Properties*** The specific gravity of the fluid is given to be SG = 0.85. We take the density of water to be 1000 kg/m$^3$. Then density of the liquid is obtained by multiplying its specific gravity by the density of water,

$$\rho = \text{SG} \times \rho_{H_2O} = (0.85)(1000 \text{ kg/m}^3) = 850 \text{ kg/m}^3$$

***Analysis*** (*a*) Knowing the absolute pressure, the atmospheric pressure can be determined from




$$P_{atm} = P - \rho g h$$
$$= (145 \text{ kPa}) - (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(5 \text{ m})\left(\frac{1 \text{ kPa}}{1000 \text{ N/m}^2}\right) = \mathbf{96.0 \ kPa}$$

(*b*) The absolute pressure at a depth of 5 m in the other liquid is

$$P = P_{atm} + \rho g h$$
$$= (96.0 \text{ kPa}) + (850 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(5 \text{ m})\left(\frac{1 \text{ kPa}}{1000 \text{ N/m}^2}\right)$$
$$= 137.7 \text{ kPa} \cong \mathbf{138 \ kPa}$$

***Discussion*** Note that at a given depth, the pressure in the lighter fluid is lower, as expected.

**3-13E**

**Solution** It is to be shown that 1 kgf/cm$^2$ = 14.223 psi.

***Analysis*** Noting that 1 kgf = 9.80665 N, 1 N = 0.22481 lbf, and 1 in = 2.54 cm, we have

$$1 \text{ kgf} = 9.80665 \text{ N} = (9.80665 \text{ N})\left(\frac{0.22481 \text{ lbf}}{1 \text{ N}}\right) = 2.20463 \text{ lbf}$$

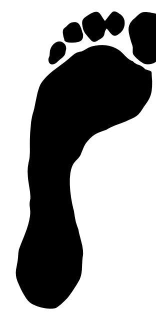
and $$1 \text{ kgf/cm}^2 = 2.20463 \text{ lbf/cm}^2 = (2.20463 \text{ lbf/cm}^2)\left(\frac{2.54 \text{ cm}}{1 \text{ in}}\right)^2 = 14.223 \text{ lbf/in}^2 = \mathbf{14.223 \ psi}$$

***Discussion*** This relationship may be used as a conversion factor.

## 3-14E

**Solution** The weight and the foot imprint area of a person are given. The pressures this man exerts on the ground when he stands on one and on both feet are to be determined.

***Assumptions*** The weight of the person is distributed uniformly on foot imprint area.



***Analysis*** The weight of the man is given to be 200 lbf. Noting that pressure is force per unit area, the pressure this man exerts on the ground is

(*a*) On one foot: $$P = \frac{W}{A} = \frac{200\ \text{lbf}}{36\ \text{in}^2} = 5.56\ \text{lbf/in}^2 = \mathbf{5.56\ psi}$$

(*a*) On both feet: $$P = \frac{W}{2A} = \frac{200\ \text{lbf}}{2\times 36\ \text{in}^2} = 2.78\ \text{lbf/in}^2 = \mathbf{2.78\ psi}$$

***Discussion*** Note that the pressure exerted on the ground (and on the feet) is reduced by half when the person stands on both feet.

## 3-15

**Solution** The mass of a woman is given. The minimum imprint area per shoe needed to enable her to walk on the snow without sinking is to be determined.

***Assumptions*** **1** The weight of the person is distributed uniformly on the imprint area of the shoes. **2** One foot carries the entire weight of a person during walking, and the shoe is sized for walking conditions (rather than standing). **3** The weight of the shoes is negligible.

***Analysis*** The mass of the woman is given to be 70 kg. For a pressure of 0.5 kPa on the snow, the imprint area of one shoe must be

$$A = \frac{W}{P} = \frac{mg}{P} = \frac{(70\ \text{kg})(9.81\ \text{m/s}^2)}{0.5\ \text{kPa}}\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1000\ \text{N/m}^2}\right) = \mathbf{1.37\ m^2}$$

***Discussion*** This is a very large area for a shoe, and such shoes would be impractical to use. Therefore, some sinking of the snow should be allowed to have shoes of reasonable size.

## 3-16

**Solution** The vacuum pressure reading of a tank is given. The absolute pressure in the tank is to be determined.

***Properties*** The density of mercury is given to be $\rho$ = 13,590 kg/m$^3$.

***Analysis*** The atmospheric (or barometric) pressure can be expressed as

$P_{abs}$

30kPa

$P_{atm}$ = 755mmHg

$$\begin{aligned} P_{atm} &= \rho g h \\ &= (13{,}590\ \text{kg/m}^3)(9.807\ \text{m/s}^2)(0.755\ \text{m})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1000\ \text{N/m}^2}\right) \\ &= 100.6\ \text{kPa} \end{aligned}$$

Then the absolute pressure in the tank becomes

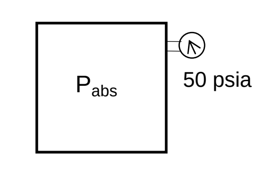
$$P_{abs} = P_{atm} - P_{vac} = 100.6 - 30 = \mathbf{70.6\ \ kPa}$$

***Discussion*** The gage pressure in the tank is the negative of the vacuum pressure, i.e., $P_{gage}$ = –30.0 kPa.

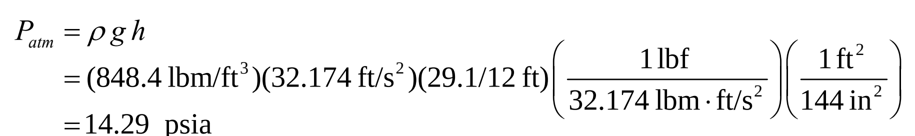
**3-17E**

**Solution** A pressure gage connected to a tank reads 50 psi. The absolute pressure in the tank is to be determined.

***Properties*** The density of mercury is given to be $\rho$ = 848.4 lbm/ft$^3$.




***Analysis*** The atmospheric (or barometric) pressure can be expressed as



$$P_{atm} = \rho g h$$
$$= (848.4 \text{ lbm/ft}^3)(32.174 \text{ ft/s}^2)(29.1/12 \text{ ft})\left(\frac{1 \text{ lbf}}{32.174 \text{ lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1 \text{ ft}^2}{144 \text{ in}^2}\right)$$
$$= 14.29 \text{ psia}$$

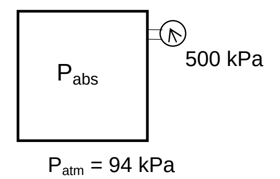
Then the absolute pressure in the tank is

$$P_{abs} = P_{gage} + P_{atm} = 50 + 14.29 = 64.29 \text{ psia} \cong \mathbf{64.3 \text{ psia}}$$

***Discussion*** This pressure is more than four times as much as standard atmospheric pressure.

**3-18**

**Solution** A pressure gage connected to a tank reads 500 kPa. The absolute pressure in the tank is to be determined.




***Analysis*** The absolute pressure in the tank is determined from

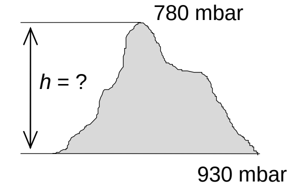
$$P_{\text{abs}} = P_{\text{gage}} + P_{\text{atm}} = 500 + 94 = \mathbf{594 \text{ kPa}}$$

***Discussion*** This pressure is almost six times greater than standard atmospheric pressure.

**3-19**

**Solution** A mountain hiker records the barometric reading before and after a hiking trip. The vertical distance climbed is to be determined.

***Assumptions*** The variation of air density and the gravitational acceleration with altitude is negligible.




***Properties*** The density of air is given to be $\rho$ = 1.20 kg/m$^3$.

***Analysis*** Taking an air column between the top and the bottom of the mountain and writing a force balance per unit base area, we obtain

$$W_{\text{air}} / A = P_{\text{bottom}} - P_{\text{top}}$$
$$(\rho g h)_{\text{air}} = P_{\text{bottom}} - P_{\text{top}}$$
$$(1.20 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(h)\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ bar}}{100{,}000 \text{ N/m}^2}\right) = (0.930 - 0.780) \text{ bar}$$

It yields $h$ = 1274 m ≅ **1270 m** (to 3 significant digits), which is also the distance climbed.

***Discussion*** A similar principle is used in some aircraft instruments to measure elevation.

**3-20**

**Solution** A barometer is used to measure the height of a building by recording reading at the bottom and at the top of the building. The height of the building is to be determined.

***Assumptions*** The variation of air density with altitude is negligible.

***Properties*** The density of air is given to be $\rho$ = 1.18 kg/m$^3$. The density of mercury is 13,600 kg/m$^3$.

***Analysis*** Atmospheric pressures at the top and at the bottom of the building are

730 mmHg

h

755 mmHg

$$P_{\text{top}} = (\rho g h)_{\text{top}}$$
$$= (13{,}600\ \text{kg/m}^3)(9.807\ \text{m/s}^2)(0.730\ \text{m})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1000\ \text{N/m}^2}\right)$$
$$= 97.36\ \text{kPa}$$

$$P_{\text{bottom}} = (\rho g h)_{\text{bottom}}$$
$$= (13{,}600\ \text{kg/m}^3)(9.807\ \text{m/s}^2)(0.755\ \text{m})\left(\frac{1\,\text{N}}{1\,\text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1000\ \text{N/m}^2}\right)$$
$$= 100.70\ \text{kPa}$$

Taking an air column between the top and the bottom of the building, we write a force balance per unit base area,

$$W_{\text{air}} / A = P_{\text{bottom}} - P_{\text{top}} \quad \text{and} \quad (\rho g h)_{\text{air}} = P_{\text{bottom}} - P_{\text{top}}$$

$$(1.18\ \text{kg/m}^3)(9.807\ \text{m/s}^2)(h)\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1000\ \text{N/m}^2}\right) = (100.70 - 97.36)\ \text{kPa}$$

which yields $h$ = 288.6 m ≅ **289 m**, which is also the height of the building.

***Discussion*** There are more accurate ways to measure the height of a building, but this method is quite simple.

---

**3-21** 


**Solution** The previous problem is reconsidered. The EES solution is to be printed out, including proper units.

***Analysis*** The EES *Equations* window is printed below, followed by the *Solution* window.

```
P_bottom=755"[mmHg]"
P_top=730"[mmHg]"
g=9.807    "[m/s^2]"    "local acceleration of gravity at sea level"
rho=1.18"[kg/m^3]"
DELTAP_abs=(P_bottom-P_top)*CONVERT('mmHg','kPa')"[kPa]"    "Delta P reading from the barometers, converted from mmHg to kPa."
DELTAP_h =rho*g*h/1000 "[kPa]"        "Equ. 1-16. Delta P due to the air fluid column height, h, between the top and bottom of the building."
"Instead of dividing by 1000 Pa/kPa we could have multiplied rho*g*h by the EES function, CONVERT('Pa','kPa')"
DELTAP_abs=DELTAP_h
```

**SOLUTION**

```
Variables in Main
DELTAP_abs=3.333 [kPa]          DELTAP_h=3.333 [kPa]
g=9.807 [m/s^2]                 h=288 [m]
P_bottom=755 [mmHg]             P_top=730 [mmHg]
rho=1.18 [kg/m^3]
```

***Discussion*** To obtain the solution in EES, simply click on the icon that looks like a calculator, or Calculate-Solve.

---

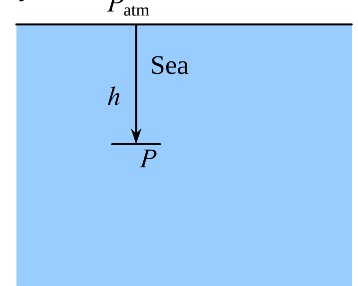
**3-22**

**Solution** A diver is moving at a specified depth from the water surface. The pressure exerted on the surface of the diver by the water is to be determined.

***Assumptions*** The variation of the density of water with depth is negligible.

***Properties*** The specific gravity of sea water is given to be SG = 1.03. We take the density of water to be 1000 kg/m$^3$.

***Analysis*** The density of the sea water is obtained by multiplying its specific gravity by the density of water which is taken to be 1000 kg/m$^3$:




$$\rho = \text{SG} \times \rho_{H_2O} = (1.03)(1000\ \text{kg/m}^3) = 1030\ \text{kg/m}^3$$

The pressure exerted on a diver at 30 m below the free surface of the sea is the absolute pressure at that location:

$$\begin{aligned} P &= P_{atm} + \rho g h \\ &= (101\,\text{kPa}) + (1030\ \text{kg/m}^3)(9.807\ \text{m/s}^2)(30\ \text{m})\left(\frac{1\,\text{kPa}}{1000\ \text{N/m}^2}\right) \\ &= \mathbf{404\ kPa} \end{aligned}$$

***Discussion*** This is about 4 times the normal sea level value of atmospheric pressure.

---

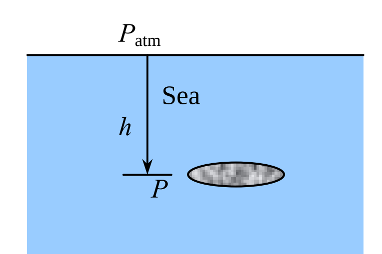
**3-23E**

**Solution** A submarine is cruising at a specified depth from the water surface. The pressure exerted on the surface of the submarine by water is to be determined.

***Assumptions*** The variation of the density of water with depth is negligible.

***Properties*** The specific gravity of sea water is given to be SG = 1.03. The density of water at 32°F is 62.4 lbm/ft$^3$.

***Analysis*** The density of the seawater is obtained by multiplying its specific gravity by the density of water,




$$\rho = \text{SG} \times \rho_{H_2O} = (1.03)(62.4\,\text{lbm/ft}^3) = 64.27\,\text{lbm/ft}^3$$



The pressure exerted on the surface of the submarine cruising 300 ft below the free surface of the sea is the absolute pressure at that location:

$$\begin{aligned} P &= P_{atm} + \rho g h \\ &= (14.7\ \text{psia}) + (64.27\ \text{lbm/ft}^3)(32.174\ \text{ft/s}^2)(300\ \text{ft})\left(\frac{1\ \text{lbf}}{32.174\ \text{lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1\ \text{ft}^2}{144\ \text{in}^2}\right) \\ &= 148.6\ \text{psia} \cong \mathbf{149\ psia} \end{aligned}$$

where we have rounded the final answer to three significant digits.

***Discussion*** This is more than 10 times the value of atmospheric pressure at sea level.

---

**3-24**

**Solution** A gas contained in a vertical piston-cylinder device is pressurized by a spring and by the weight of the piston. The pressure of the gas is to be determined.

***Analysis*** Drawing the free body diagram of the piston and balancing the vertical forces yields

$$PA = P_{atm}A + W + F_{spring}$$

Thus,

$$P = P_{\text{atm}} + \frac{mg + F_{\text{spring}}}{A}$$

$$= (95 \text{ kPa}) + \frac{(4 \text{ kg})(9.807 \text{ m/s}^2) + 60 \text{ N}}{35\times10^{-4} \text{ m}^2}\left(\frac{1 \text{ kPa}}{1000 \text{ N/m}^2}\right) = 123.4 \text{ kPa} \cong \mathbf{123 \text{ kPa}}$$

***Discussion*** This setup represents a crude but functional way to control the pressure in a tank.

---

**3-25** 

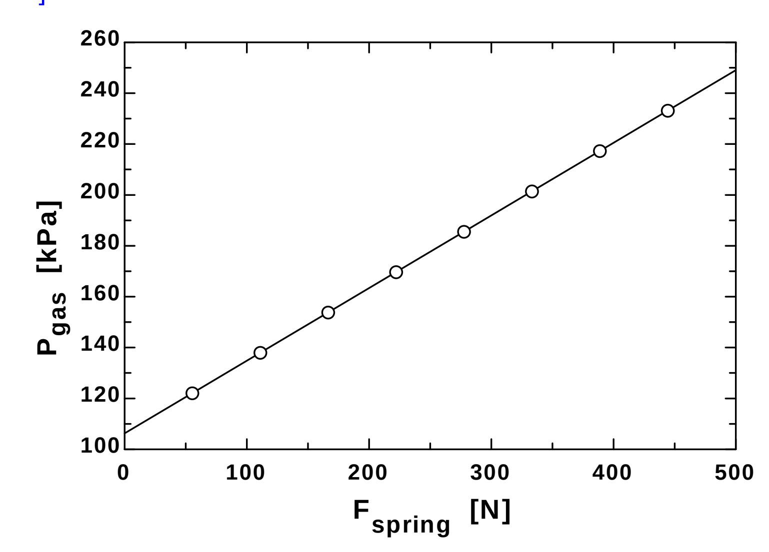
**Solution** The previous problem is reconsidered. The effect of the spring force in the range of 0 to 500 N on the pressure inside the cylinder is to be investigated. The pressure against the spring force is to be plotted, and results are to be discussed.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
g=9.807"[m/s^2]"
P_atm= 95"[kPa]"
m_piston=4"[kg]"
{F_spring=60"[N]"}
A=35*CONVERT('cm^2','m^2')"[m^2]"
W_piston=m_piston*g"[N]"
F_atm=P_atm*A*CONVERT('kPa','N/m^2')"[N]"
"From the free body diagram of the piston, the balancing vertical forces yield:"
F_gas= F_atm+F_spring+W_piston"[N]"
P_gas=F_gas/A*CONVERT('N/m^2','kPa')"[kPa]"
```

*Results*:

| $F_{spring}$ [N] | $P_{gas}$ [kPa] |
|---|---|
| 0 | 106.2 |
| 55.56 | 122.1 |
| 111.1 | 138 |
| 166.7 | 153.8 |
| 222.2 | 169.7 |
| 277.8 | 185.6 |
| 333.3 | 201.4 |
| 388.9 | 217.3 |
| 444.4 | 233.2 |
| 500 | 249.1 |




***Discussion*** The relationship is linear, as expected.

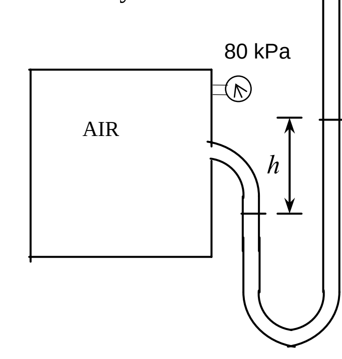
---

**3-26** **[*Also solved using EES on enclosed DVD*]**

**Solution** Both a pressure gage and a manometer are attached to a tank of gas to measure its pressure. For a specified reading of gage pressure, the difference between the fluid levels of the two arms of the manometer is to be determined for mercury and water.

***Properties*** The densities of water and mercury are given to be $\rho_{\text{water}}$ = 1000 kg/m$^3$ and be $\rho_{\text{Hg}}$ = 13,600 kg/m$^3$.

***Analysis*** The gage pressure is related to the vertical distance $h$ between the two fluid levels by




$$P_{\text{gage}} = \rho g h \quad \longrightarrow \quad h = \frac{P_{\text{gage}}}{\rho g}$$

(*a*) For mercury,

$$h = \frac{P_{\text{gage}}}{\rho_{Hg} g} = \frac{80 \text{ kPa}}{(13600 \text{ kg/m}^3)(9.807 \text{ m/s}^2)}\left(\frac{1 \text{ kN/m}^2}{1 \text{ kPa}}\right)\left(\frac{1000 \text{ kg/m}\cdot\text{s}^2}{1 \text{ kN}}\right) = \mathbf{0.60 \text{ m}}$$

(*b*) For water,

$$h = \frac{P_{\text{gage}}}{\rho_{H_2O} g} = \frac{80 \text{ kPa}}{(1000 \text{ kg/m}^3)(9.807 \text{m/s}^2)}\left(\frac{1 \text{ kN/m}^2}{1 \text{ kPa}}\right)\left(\frac{1000 \text{ kg/m}\cdot\text{s}^2}{1 \text{ kN}}\right) = \mathbf{8.16 \text{ m}}$$

***Discussion*** The manometer with water is more precise since the column height is bigger (better resolution). However, a column of water more than 8 meters high would be impractical, so mercury is the better choice of manometer fluid here.


**3-27** 

**Solution** The previous problem is reconsidered. The effect of the manometer fluid density in the range of 800 to 13,000 kg/m$^3$ on the differential fluid height of the manometer is to be investigated. Differential fluid height is to be plotted as a function of the density, and the results are to be discussed.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
Function fluid_density(Fluid$)
    If fluid$='Mercury' then  fluid_density=13600 else fluid_density=1000
end

{Input from the diagram window.  If the diagram window is hidden, then all of the input must come from the
equations window.  Also note that brackets can also denote comments - but these comments do not appear
in the formatted equations window.}

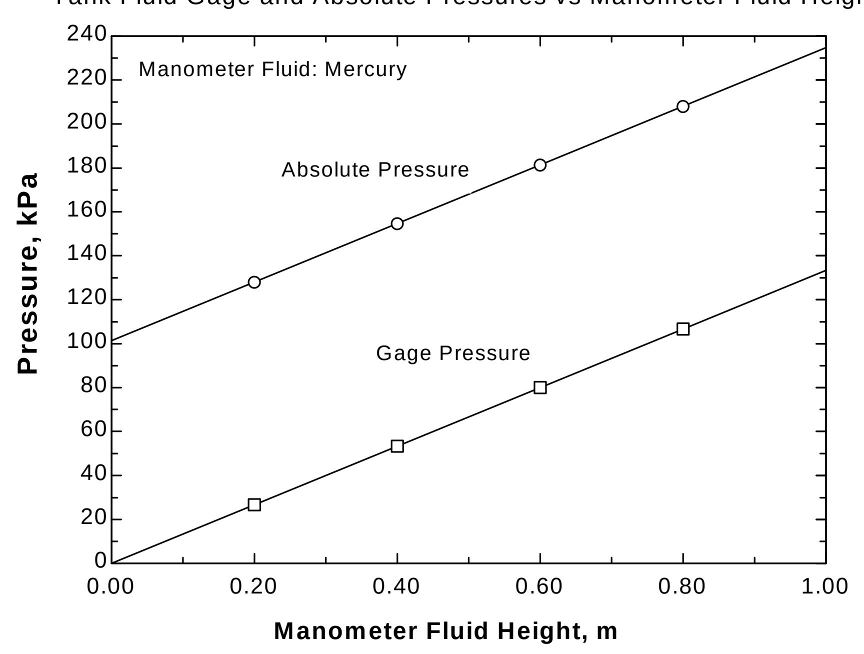
{Fluid$='Mercury'
P_atm = 101.325                    "kpa"
DELTAP=80            "kPa  Note how DELTAP is displayed on the Formatted Equations Window."}

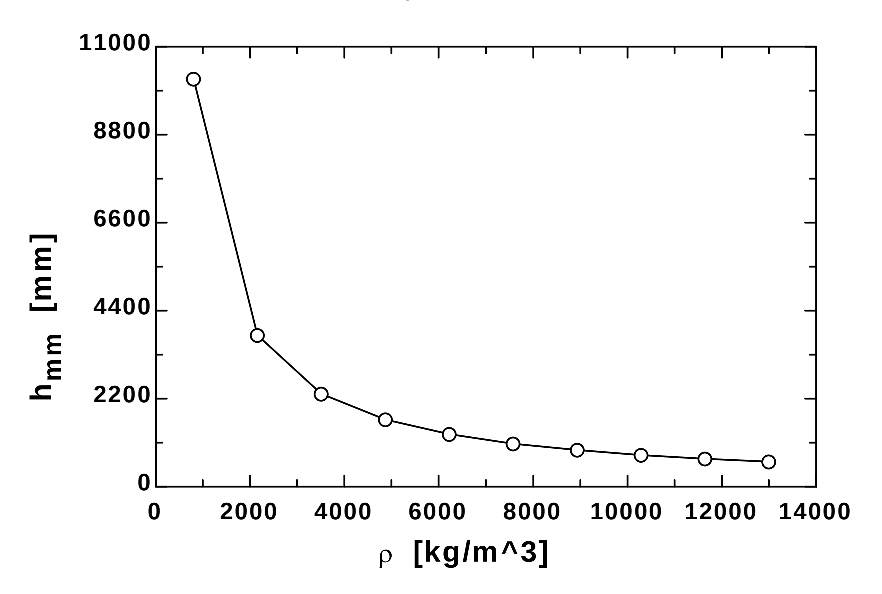
g=9.807                             "m/s2, local acceleration of gravity at sea level"
rho=Fluid_density(Fluid$)       "Get the fluid density, either Hg or H2O, from the function"
"To plot fluid height against density place {} around the above equation.  Then set up the parametric table
and solve."
DELTAP = RHO*g*h/1000
"Instead of dividing by 1000 Pa/kPa we could have multiplied by the EES function, CONVERT('Pa','kPa')"
h_mm=h*convert('m','mm')       "The fluid height in mm is found using the built-in CONVERT function."
P_abs= P_atm + DELTAP

"To make the graph, hide the diagram window and remove the {}brackets from Fluid$ and from P_atm.
Select New Parametric Table from the Tables menu.  Choose P_abs, DELTAP and h to be in the table.
Choose Alter Values from the Tables menu.  Set values of h to range from 0 to 1 in steps of 0.2.  Choose
Solve Table (or press F3) from the Calculate menu.  Choose New Plot Window from the Plot menu.
Choose to plot P_abs vs h and then choose Overlay Plot from the Plot menu and plot DELTAP on the same
scale."
```

*Results*:

| $h_{mm}$ [mm] | $\rho$ [kg/m$^3$] |
|---|---|
| 10197 | 800 |
| 3784 | 2156 |
| 2323 | 3511 |
| 1676 | 4867 |
| 1311 | 6222 |
| 1076 | 7578 |
| 913.1 | 8933 |
| 792.8 | 10289 |
| 700.5 | 11644 |
| 627.5 | 13000 |






***Discussion*** Many comments are provided in the Equation window above to help you learn some of the features of EES.

---

**3-28**

**Solution** The air pressure in a tank is measured by an oil manometer. For a given oil-level difference between the two columns, the absolute pressure in the tank is to be determined.

AIR

0.45 m

$P_{atm}$ = 98 kPa

***Properties*** The density of oil is given to be $\rho$ = 850 kg/m$^3$.

***Analysis*** The absolute pressure in the tank is determined from

$$P = P_{atm} + \rho g h$$

$$= (98\ \text{kPa}) + (850\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(0.45\,\text{m})\left(\frac{1\ \text{kPa}}{1000\ \text{N/m}^2}\right)$$

$$= 101.75\ \text{kPa} \cong \mathbf{102\ kPa}$$

***Discussion*** If a heavier liquid, such as water, were used for the manometer fluid, the column height would be smaller, and thus the reading would be less precise (lower resolution).

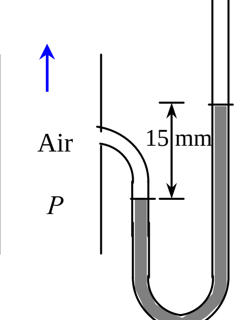
---

**3-29**

**Solution** The air pressure in a duct is measured by a mercury manometer. For a given mercury-level difference between the two columns, the absolute pressure in the duct is to be determined.

***Properties*** The density of mercury is given to be $\rho$ = 13,600 kg/m$^3$.

***Analysis*** (*a*) The pressure in the duct is above atmospheric pressure since the fluid column on the duct side is at a lower level.




(*b*) The absolute pressure in the duct is determined from

$$P = P_{atm} + \rho gh$$
$$= (100 \text{ kPa}) + (13{,}600 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(0.015 \text{ m})\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ kPa}}{1000 \text{ N/m}^2}\right)$$
$$= 102.00 \text{ kPa} \cong \mathbf{102 \text{ kPa}}$$

***Discussion*** When measuring pressures in a fluid flow, the *difference* between two pressures is usually desired. In this case, the difference is between the measurement point and atmospheric pressure.

---

**3-30**

**Solution** The air pressure in a duct is measured by a mercury manometer. For a given mercury-level difference between the two columns, the absolute pressure in the duct is to be determined.

***Properties*** The density of mercury is given to be $\rho$ = 13,600 kg/m$^3$.

***Analysis*** (*a*) The pressure in the duct is above atmospheric pressure since the fluid column on the duct side is at a lower level.

(*b*) The absolute pressure in the duct is determined from

$$P = P_{atm} + \rho gh$$
$$= (100 \text{ kPa}) + (13{,}600 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(0.030 \text{ m})\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ kPa}}{1000 \text{ N/m}^2}\right)$$
$$= 104.00 \text{ kPa} \cong \mathbf{104 \text{ kPa}}$$

***Discussion*** The final result is given to three significant digits.

---

**3-31**

**Solution** The systolic and diastolic pressures of a healthy person are given in mm of Hg. These pressures are to be expressed in kPa, psi, and meters of water column.

***Assumptions*** Both mercury and water are incompressible substances.

***Properties*** We take the densities of water and mercury to be 1000 kg/m$^3$ and 13,600 kg/m$^3$, respectively.

***Analysis*** Using the relation $P = \rho g h$ for gage pressure, the high and low pressures are expressed as

$$P_{\text{high}} = \rho g h_{\text{high}} = (13{,}600\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(0.12\ \text{m})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1000\ \text{N/m}^2}\right) = \mathbf{16.0\ kPa}$$

$$P_{\text{low}} = \rho g h_{\text{low}} = (13{,}600\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(0.08\ \text{m})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1000\ \text{N/m}^2}\right) = \mathbf{10.7\ kPa}$$

Noting that 1 psi = 6.895 kPa,

$$P_{\text{high}} = (16.0\ \text{kPa})\left(\frac{1\ \text{psi}}{6.895\ \text{kPa}}\right) = \mathbf{2.32\ psi} \quad \text{and} \quad P_{\text{low}} = (10.7\ \text{kPa})\left(\frac{1\ \text{psi}}{6.895\ \text{kPa}}\right) = \mathbf{1.55\ psi}$$

For a given pressure, the relation $P = \rho g h$ is expressed for mercury and water as $P = \rho_{\text{water}} g h_{\text{water}}$ and $P = \rho_{\text{mercury}} g h_{\text{mercury}}$. Setting these two relations equal to each other and solving for water height gives

$$P = \rho_{\text{water}} g h_{\text{water}} = \rho_{\text{mercury}} g h_{\text{mercury}} \quad \rightarrow \quad h_{\text{water}} = \frac{\rho_{\text{mercury}}}{\rho_{\text{water}}} h_{\text{mercury}}$$

Therefore,

$$h_{\text{water, high}} = \frac{\rho_{\text{mercury}}}{\rho_{\text{water}}} h_{\text{mercury, high}} = \frac{13{,}600\ \text{kg/m}^3}{1000\ \text{kg/m}^3}(0.12\ \text{m}) = \mathbf{1.63\ m}$$

$$h_{\text{water, low}} = \frac{\rho_{\text{mercury}}}{\rho_{\text{water}}} h_{\text{mercury, low}} = \frac{13{,}600\ \text{kg/m}^3}{1000\ \text{kg/m}^3}(0.08\ \text{m}) = \mathbf{1.09\ m}$$

***Discussion*** Note that measuring blood pressure with a water monometer would involve water column heights higher than the person's height, and thus it is impractical. This problem shows why mercury is a suitable fluid for blood pressure measurement devices.

---

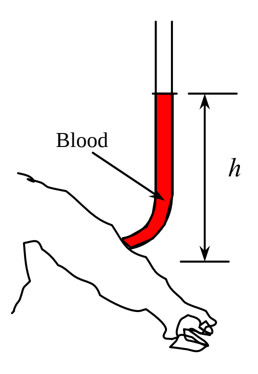


**3-32**

**Solution** A vertical tube open to the atmosphere is connected to the vein in the arm of a person. The height that the blood rises in the tube is to be determined.

***Assumptions*** **1** The density of blood is constant. **2** The gage pressure of blood is 120 mmHg.

***Properties*** The density of blood is given to be $\rho$ = 1050 kg/m$^3$.




***Analysis*** For a given gage pressure, the relation $P = \rho g h$ can be expressed for mercury and blood as $P = \rho_{\text{blood}} g h_{\text{blood}}$ and $P = \rho_{\text{mercury}} g h_{\text{mercury}}$. Setting these two relations equal to each other we get

$$P = \rho_{\text{blood}} g h_{\text{blood}} = \rho_{\text{mercury}} g h_{\text{mercury}}$$

Solving for blood height and substituting gives

$$h_{\text{blood}} = \frac{\rho_{\text{mercury}}}{\rho_{\text{blood}}} h_{\text{mercury}} = \frac{13{,}600\ \text{kg/m}^3}{1050\ \text{kg/m}^3}(0.12\ \text{m}) = \mathbf{1.55\ m}$$

***Discussion*** Note that the blood can rise about one and a half meters in a tube connected to the vein. This explains why IV tubes must be placed high to force a fluid into the vein of a patient.

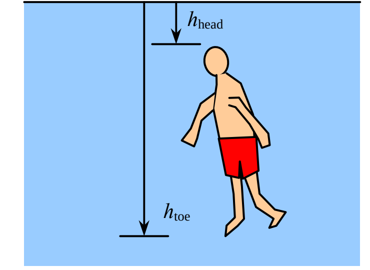
---

**3-33**

**Solution** A man is standing in water vertically while being completely submerged. The difference between the pressure acting on his head and the pressure acting on his toes is to be determined.

***Assumptions*** Water is an incompressible substance, and thus the density does not change with depth.

***Properties*** We take the density of water to be $\rho$ =1000 kg/m$^3$.




***Analysis*** The pressures at the head and toes of the person can be expressed as

$$P_{head} = P_{atm} + \rho g h_{head} \quad \text{and} \quad P_{toe} = P_{atm} + \rho g h_{toe}$$

where $h$ is the vertical distance of the location in water from the free surface. The pressure difference between the toes and the head is determined by subtracting the first relation above from the second,

$$P_{toe} - P_{head} = \rho g h_{toe} - \rho g h_{head} = \rho g (h_{toe} - h_{head})$$

Substituting,

$$P_{toe} - P_{head} = (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(1.80 \text{ m - 0})\left(\frac{1 \text{N}}{1 \text{kg} \cdot \text{m/s}^2}\right)\left(\frac{1 \text{kPa}}{1000 \text{ N/m}^2}\right) = \mathbf{17.7 \text{ kPa}}$$

***Discussion*** This problem can also be solved by noting that the atmospheric pressure (1 atm = 101.325 kPa) is equivalent to 10.3-m of water height, and finding the pressure that corresponds to a water height of 1.8 m.

---

**3-34**

**Solution** Water is poured into the U-tube from one arm and oil from the other arm. The water column height in one arm and the ratio of the heights of the two fluids in the other arm are given. The height of each fluid in that arm is to be determined.

***Assumptions*** Both water and oil are incompressible substances.

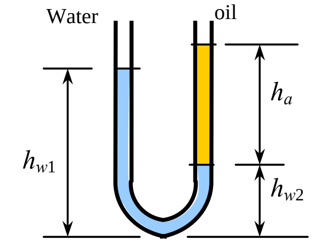
***Properties*** The density of oil is given to be $\rho_{oil}$ = 790 kg/m$^3$. We take the density of water to be $\rho_w$ =1000 kg/m$^3$.

***Analysis*** The height of water column in the left arm of the manometer is given to be $h_{w1}$ = 0.70 m. We let the height of water and oil in the right arm to be $h_{w2}$ and $h_a$, respectively. Then, $h_a = 6h_{w2}$. Noting that both arms are open to the atmosphere, the pressure at the bottom of the U-tube can be expressed as

$$P_{bottom} = P_{atm} + \rho_w g h_{w1} \quad \text{and} \quad P_{bottom} = P_{atm} + \rho_w g h_{w2} + \rho_a g h_a$$

Setting them equal to each other and simplifying,

$$\rho_w g h_{w1} = \rho_w g h_{w2} + \rho_a g h_a \quad \rightarrow \quad \rho_w h_{w1} = \rho_w h_{w2} + \rho_a h_a \quad \rightarrow \quad h_{w1} = h_{w2} + (\rho_a / \rho_w) h_a$$




Noting that $h_a = 6h_{w2}$ and we take $\rho_a = \rho_{oil}$, the water and oil column heights in the second arm are determined to be

$$0.7 \text{ m} = h_{w2} + (790/1000)6h_{w2} \quad \rightarrow \quad h_{w2} = \mathbf{0.122 \text{ m}}$$

$$0.7 \text{ m} = 0.122 \text{ m} + (790/1000)h_a \quad \rightarrow \quad h_a = \mathbf{0.732 \text{ m}}$$

***Discussion*** Note that the fluid height in the arm that contains oil is higher. This is expected since oil is lighter than water.

---

**3-35**

**Solution** The hydraulic lift in a car repair shop is to lift cars. The fluid gage pressure that must be maintained in the reservoir is to be determined.

***Assumptions*** The weight of the piston of the lift is negligible.

***Analysis*** Pressure is force per unit area, and thus the gage pressure required is simply the ratio of the weight of the car to the area of the lift,

$W = mg$, $P_{atm}$, $P$

$$P_{gage} = \frac{W}{A} = \frac{mg}{\pi D^2/4} = \frac{(2000\text{ kg})(9.81\text{ m/s}^2)}{\pi(0.30\text{ m})^2/4}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 278\text{ kN/m}^2 = \mathbf{278\ kPa}$$

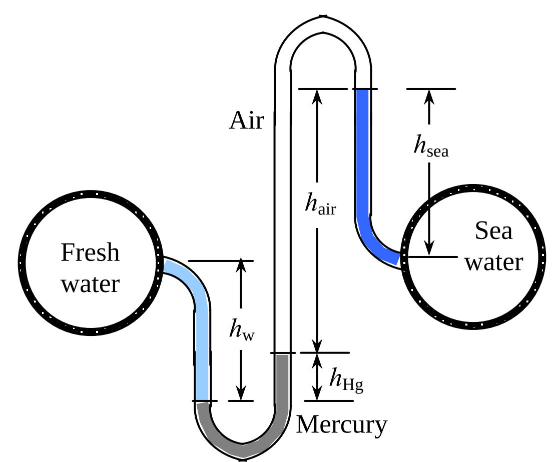
***Discussion*** Note that the pressure level in the reservoir can be reduced by using a piston with a larger area.

---



**3-36**

**Solution** Fresh and seawater flowing in parallel horizontal pipelines are connected to each other by a double U-tube manometer. The pressure difference between the two pipelines is to be determined.




***Assumptions*** **1** All the liquids are incompressible. **2** The effect of air column on pressure is negligible.

***Properties*** The densities of seawater and mercury are given to be $\rho_{sea}$ = 1035 kg/m$^3$ and $\rho_{Hg}$ = 13,600 kg/m$^3$. We take the density of water to be $\rho_w$ =1000 kg/m$^3$.

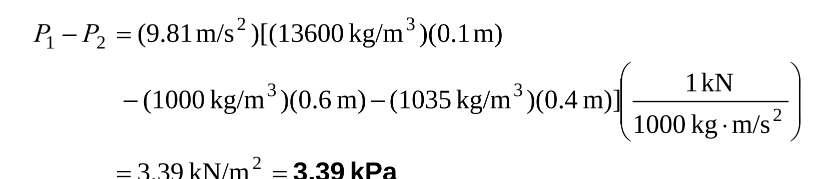
***Analysis*** Starting with the pressure in the fresh water pipe (point 1) and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the sea water pipe (point 2), and setting the result equal to $P_2$ gives

$$P_1 + \rho_w gh_w - \rho_{Hg} gh_{Hg} - \rho_{air} gh_{air} + \rho_{sea} gh_{sea} = P_2$$

Rearranging and neglecting the effect of air column on pressure,

$$P_1 - P_2 = -\rho_w gh_w + \rho_{Hg} gh_{Hg} - \rho_{sea} gh_{sea} = g(\rho_{Hg} h_{Hg} - \rho_w h_w - \rho_{sea} h_{sea})$$

Substituting,



$$P_1 - P_2 = (9.81\text{ m/s}^2)[(13600\text{ kg/m}^3)(0.1\text{ m})$$
$$-(1000\text{ kg/m}^3)(0.6\text{ m}) - (1035\text{ kg/m}^3)(0.4\text{ m})]\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)$$
$$= 3.39\text{ kN/m}^2 = \mathbf{3.39\ kPa}$$

Therefore, the pressure in the fresh water pipe is 3.39 kPa higher than the pressure in the sea water pipe.

***Discussion*** A 0.70-m high air column with a density of 1.2 kg/m$^3$ corresponds to a pressure difference of 0.008 kPa. Therefore, its effect on the pressure difference between the two pipes is negligible.

---

**3-37**

**Solution** Fresh and seawater flowing in parallel horizontal pipelines are connected to each other by a double U-tube manometer. The pressure difference between the two pipelines is to be determined.

***Assumptions*** All the liquids are incompressible.

***Properties*** The densities of seawater and mercury are given to be $\rho_{sea}$ = 1035 kg/m$^3$ and $\rho_{Hg}$ = 13,600 kg/m$^3$. We take the density of water to be $\rho_w$ =1000 kg/m$^3$. The specific gravity of oil is given to be 0.72, and thus its density is 720 kg/m$^3$.

***Analysis*** Starting with the pressure in the fresh water pipe (point 1) and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the sea water pipe (point 2), and setting the result equal to $P_2$ gives

$$P_1 + \rho_w gh_w - \rho_{Hg} gh_{Hg} - \rho_{oil} gh_{oil} + \rho_{sea} gh_{sea} = P_2$$

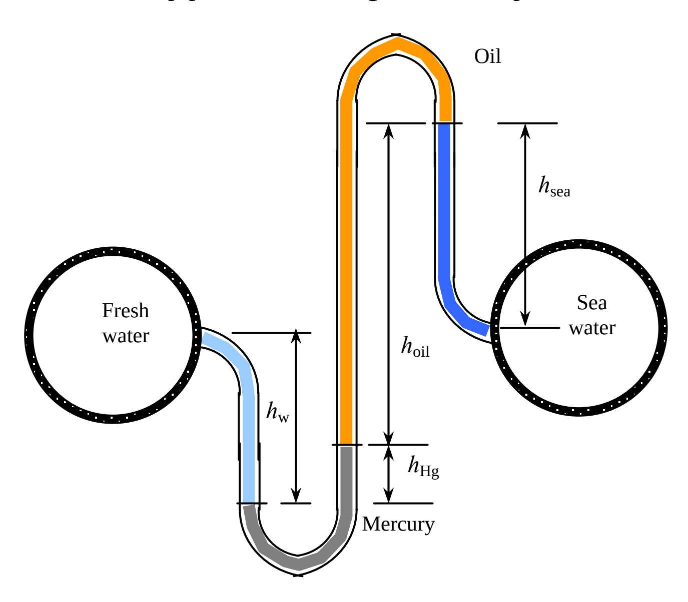
Rearranging,

$$\begin{aligned} P_1 - P_2 &= -\rho_w gh_w + \rho_{Hg} gh_{Hg} + \rho_{oil} gh_{oil} - \rho_{sea} gh_{sea} \\ &= g(\rho_{Hg} h_{Hg} + \rho_{oil} h_{oil} - \rho_w h_w - \rho_{sea} h_{sea}) \end{aligned}$$

Substituting,

$$\begin{aligned} P_1 - P_2 &= (9.81\,\text{m/s}^2)[(13600\,\text{kg/m}^3)(0.1\,\text{m}) + (720\,\text{kg/m}^3)(0.7\,\text{m}) - (1000\,\text{kg/m}^3)(0.6\,\text{m}) \\ &\quad - (1035\,\text{kg/m}^3)(0.4\,\text{m})]\left(\frac{1\,\text{kN}}{1000\,\text{kg}\cdot\text{m/s}^2}\right) \\ &= 8.34\,\text{kN/m}^2 = \mathbf{8.34\,kPa} \end{aligned}$$

Therefore, the pressure in the fresh water pipe is 8.34 kPa higher than the pressure in the sea water pipe.




***Discussion*** The result is greater than that of the previous problem since the oil is heavier than the air.

**3-38E**

**Solution** The pressure in a natural gas pipeline is measured by a double U-tube manometer with one of the arms open to the atmosphere. The absolute pressure in the pipeline is to be determined.

***Assumptions*** **1** All the liquids are incompressible. **2** The effect of air column on pressure is negligible. **3** The pressure throughout the natural gas (including the tube) is uniform since its density is low.

***Properties*** We take the density of water to be $\rho_w$ = 62.4 lbm/ft$^3$. The specific gravity of mercury is given to be 13.6, and thus its density is $\rho_{Hg}$ = 13.6×62.4 = 848.6 lbm/ft$^3$.

***Analysis*** Starting with the pressure at point 1 in the natural gas pipeline, and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the free surface of oil where the oil tube is exposed to the atmosphere, and setting the result equal to $P_{atm}$ gives

$$P_1 - \rho_{Hg} g h_{Hg} - \rho_{water} g h_{water} = P_{atm}$$

Solving for $P_{1,}$

$$P_1 = P_{atm} + \rho_{Hg} g h_{Hg} + \rho_{water} g h_1$$

Substituting,

$$P = 14.2 \text{ psia} + (32.2 \text{ ft/s}^2)[(848.6 \text{ lbm/ft}^3)(6/12 \text{ ft}) + (62.4 \text{lbm/ft}^3)(27/12 \text{ ft})]\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1 \text{ ft}^2}{144 \text{ in}^2}\right)$$
$$= \mathbf{18.1 \text{ psia}}$$



***Discussion*** Note that jumping horizontally from one tube to the next and realizing that pressure remains the same in the same fluid simplifies the analysis greatly. Also, it can be shown that the 15-in high air column with a density of 0.075 lbm/ft$^3$ corresponds to a pressure difference of 0.00065 psi. Therefore, its effect on the pressure difference between the two pipes is negligible.

**3-39E**

**Solution** The pressure in a natural gas pipeline is measured by a double U-tube manometer with one of the arms open to the atmosphere. The absolute pressure in the pipeline is to be determined.

***Assumptions*** **1** All the liquids are incompressible. **2** The pressure throughout the natural gas (including the tube) is uniform since its density is low.

***Properties*** We take the density of water to be $\rho_w$ = 62.4 lbm/ft$^3$. The specific gravity of mercury is given to be 13.6, and thus its density is $\rho_{Hg}$ = 13.6×62.4 = 848.6 lbm/ft$^3$. The specific gravity of oil is given to be 0.69, and thus its density is $\rho_{oil}$ = 0.69×62.4 = 43.1 lbm/ft$^3$.

***Analysis*** Starting with the pressure at point 1 in the natural gas pipeline, and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the free surface of oil where the oil tube is exposed to the atmosphere, and setting the result equal to $P_{atm}$ gives

$$P_1 - \rho_{Hg} g h_{Hg} + \rho_{oil} g h_{oil} - \rho_{water} g h_{water} = P_{atm}$$

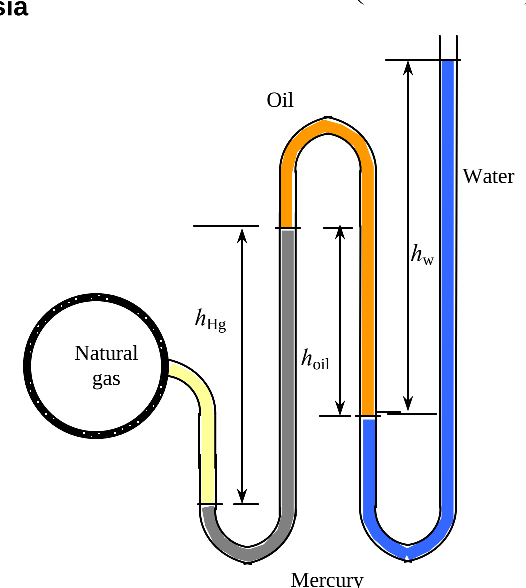
Solving for $P_{1,}$

$$P_1 = P_{atm} + \rho_{Hg} g h_{Hg} + \rho_{water} g h_1 - \rho_{oil} g h_{oil}$$

Substituting,

$$P_1 = 14.2\,\text{psia} + (32.2\ \text{ft/s}^2)[(848.6\,\text{lbm/ft}^3)(6/12\ \text{ft}) + (62.4\,\text{lbm/ft}^3)(27/12\ \text{ft}) - (43.1\,\text{lbm/ft}^3)(15/12\ \text{ft})]\left(\frac{1\,\text{lbf}}{32.2\,\text{lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1\,\text{ft}^2}{144\,\text{in}^2}\right)$$

$$= \mathbf{17.7\,psia}$$




***Discussion*** Note that jumping horizontally from one tube to the next and realizing that pressure remains the same in the same fluid simplifies the analysis greatly.

**3-40**

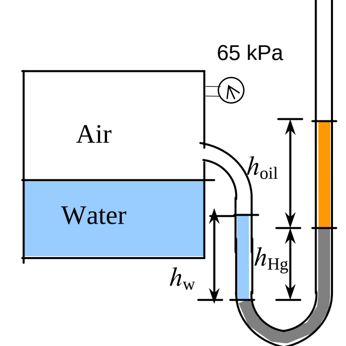
**Solution** The gage pressure of air in a pressurized water tank is measured simultaneously by both a pressure gage and a manometer. The differential height $h$ of the mercury column is to be determined.

***Assumptions*** The air pressure in the tank is uniform (i.e., its variation with elevation is negligible due to its low density), and thus the pressure at the air-water interface is the same as the indicated gage pressure.

***Properties*** We take the density of water to be $\rho_w$ =1000 kg/m$^3$. The specific gravities of oil and mercury are given to be 0.72 and 13.6, respectively.

***Analysis*** Starting with the pressure of air in the tank (point 1), and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the free surface of oil where the oil tube is exposed to the atmosphere, and setting the result equal to $P_{atm}$ gives

$$P_1 + \rho_w g h_w - \rho_{Hg} g h_{Hg} - \rho_{oil} g h_{oil} = P_{atm}$$




Rearranging,

$$P_1 - P_{atm} = \rho_{oil} g h_{oil} + \rho_{Hg} g h_{Hg} - \rho_w g h_w$$

or,

$$\frac{P_{1,gage}}{\rho_w g} = \rho_{s,oil} h_{oil} + \rho_{s,Hg} h_{Hg} - h_w$$

Substituting,

$$\left(\frac{65\,\text{kPa}}{(1000\,\text{kg/m}^3)(9.81\,\text{m/s}^2)}\right)\left(\frac{1000\ \text{kg}\cdot\text{m/s}^2}{1\,\text{kPa.}\cdot\text{m}^2}\right) = 0.72\times(0.75\,\text{m}) + 13.6\times h_{Hg} - 0.3\,\text{m}$$

Solving for $h_{Hg}$ gives $h_{Hg}$ = **0.47 m**. Therefore, the differential height of the mercury column must be 47 cm.

***Discussion*** Double instrumentation like this allows one to verify the measurement of one of the instruments by the measurement of another instrument.

---

**3-41**

**Solution** The gage pressure of air in a pressurized water tank is measured simultaneously by both a pressure gage and a manometer. The differential height $h$ of the mercury column is to be determined.

***Assumptions*** The air pressure in the tank is uniform (i.e., its variation with elevation is negligible due to its low density), and thus the pressure at the air-water interface is the same as the indicated gage pressure.

***Properties*** We take the density of water to be $\rho_w$ =1000 kg/m$^3$. The specific gravities of oil and mercury are given to be 0.72 and 13.6, respectively.

***Analysis*** Starting with the pressure of air in the tank (point 1), and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the free surface of oil where the oil tube is exposed to the atmosphere, and setting the result equal to $P_{atm}$ gives

$$P_1 + \rho_w g h_w - \rho_{Hg} g h_{Hg} - \rho_{oil} g h_{oil} = P_{atm}$$

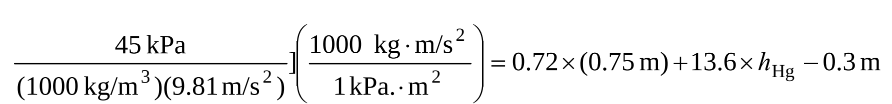
Rearranging,

$$P_1 - P_{atm} = \rho_{oil} g h_{oil} + \rho_{Hg} g h_{Hg} - \rho_w g h_w$$

or,

$$\frac{P_{1,gage}}{\rho_w g} = SG_{oil} h_{oil} + SG_{Hg} h_{Hg} - h_w$$

Substituting,



$$\frac{45\text{ kPa}}{(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)}\left(\frac{1000\text{ kg}\cdot\text{m/s}^2}{1\text{ kPa.}\cdot\text{m}^2}\right) = 0.72\times(0.75\text{ m}) + 13.6\times h_{Hg} - 0.3\text{ m}$$



Solving for $h_{Hg}$ gives $h_{Hg}$ = **0.32 m**. Therefore, **the differential height of the mercury column must be 32 cm**.

***Discussion*** Double instrumentation like this allows one to verify the measurement of one of the instruments by the measurement of another instrument.

---

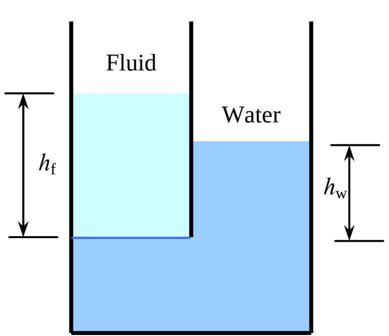


**3-42**

**Solution** The top part of a water tank is divided into two compartments, and a fluid with an unknown density is poured into one side. The levels of the water and the liquid are measured. The density of the fluid is to be determined.

***Assumptions*** **1** Both water and the added liquid are incompressible substances. **2** The added liquid does not mix with water.

***Properties*** We take the density of water to be $\rho$ =1000 kg/m$^3$.




***Analysis*** Both fluids are open to the atmosphere. Noting that the pressure of both water and the added fluid is the same at the contact surface, the pressure at this surface can be expressed as

$$P_{contact} = P_{atm} + \rho_f g h_f = P_{atm} + \rho_w g h_w$$

Simplifying, we have $\rho_f g h_f = \rho_w g h_w$ . Solving for $\rho_f$ gives

$$\rho_f = \frac{h_w}{h_f}\rho_w = \frac{45\text{ cm}}{80\text{ cm}}\left(1000\text{ kg/m}^3\right) = 562.5\text{ kg/m}^3 \cong \mathbf{563\ kg/m^3}$$

***Discussion*** Note that the added fluid is lighter than water as expected (a heavier fluid would sink in water).

---

**3-43**

**Solution** A load on a hydraulic lift is to be raised by pouring oil from a thin tube. The height of oil in the tube required in order to raise that weight is to be determined.

***Assumptions*** **1** The cylinders of the lift are vertical. **2** There are no leaks. **3** Atmospheric pressure act on both sides, and thus it can be disregarded.

***Properties*** The density of oil is given to be $\rho$ =780 kg/m$^3$.

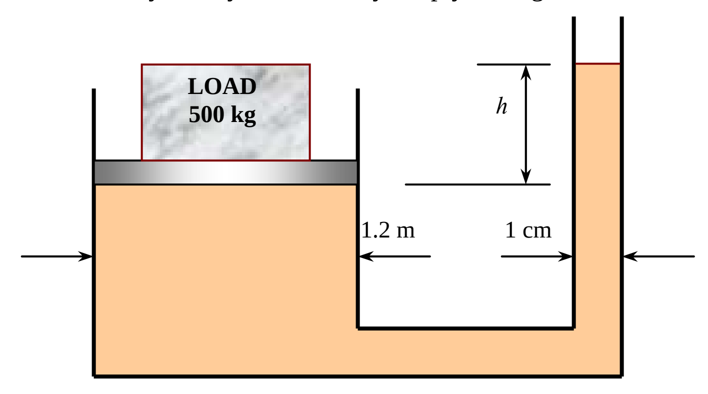
***Analysis*** Noting that pressure is force per unit area, the gage pressure in the fluid under the load is simply the ratio of the weight to the area of the lift,

$$P_{\text{gage}} = \frac{W}{A} = \frac{mg}{\pi D^2/4} = \frac{(500\text{ kg})(9.81\text{ m/s}^2)}{\pi(1.20\text{ m})^2/4}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 4.34\text{ kN/m}^2 = 4.34\text{ kPa}$$

The required oil height that will cause 4.34 kPa of pressure rise is

$$P_{\text{gage}} = \rho gh \;\rightarrow\; h = \frac{P_{\text{gage}}}{\rho g} = \frac{4.34\text{ kN/m}^2}{(780\text{ kg/m}^3)(9.81\text{ m/s}^2)}\left(\frac{1000\text{ kg}\cdot\text{m/s}^2}{1\text{kN/m}^2}\right) = \mathbf{0.567\text{ m}}$$

Therefore, a 500 kg load can be raised by this hydraulic lift by simply raising the oil level in the tube by 56.7 cm.



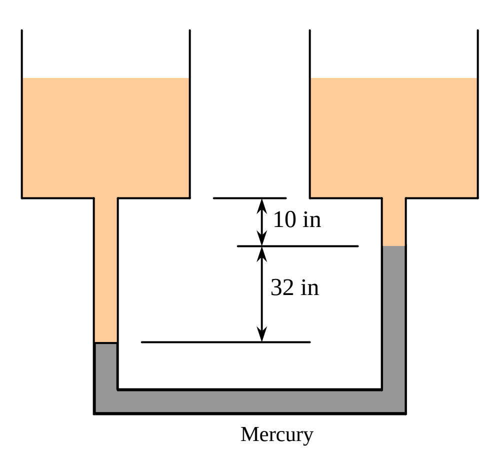

***Discussion*** Note that large weights can be raised by little effort in hydraulic lift by making use of Pascal's principle.

## 3-44E

**Solution** Two oil tanks are connected to each other through a mercury manometer. For a given differential height, the pressure difference between the two tanks is to be determined.

***Assumptions*** **1** Both the oil and mercury are incompressible fluids. **2** The oils in both tanks have the same density.

***Properties*** The densities of oil and mercury are given to be $\rho_{oil}$ = 45 lbm/ft$^3$ and $\rho_{Hg}$ = 848 lbm/ft$^3$.




***Analysis*** Starting with the pressure at the bottom of tank 1 (where pressure is $P_1$) and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the bottom of tank 2 (where pressure is $P_2$) gives

$$P_1 + \rho_{oil} g(h_1 + h_2) - \rho_{Hg} gh_2 - \rho_{oil} gh_1 = P_2$$

where $h_1$ = 10 in and $h_2$ = 32 in. Rearranging and simplifying,

$$P_1 - P_2 = \rho_{Hg} gh_2 - \rho_{oil} gh_2 = (\rho_{Hg} - \rho_{oil}) gh_2$$

Substituting,

$$\Delta P = P_1 - P_2 = (848 - 45 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)(32/12 \text{ ft})\left(\frac{1 \text{lbf}}{32.2 \text{lbm} \cdot \text{ft/s}^2}\right)\left(\frac{1 \text{ft}^2}{144 \text{in}^2}\right) = \mathbf{14.9 \ psia}$$

Therefore, the pressure in the left oil tank is 14.9 psia higher than the pressure in the right oil tank.

***Discussion*** Note that large pressure differences can be measured conveniently by mercury manometers. If a water manometer were used in this case, the differential height would be over 30 ft.

---

## 3-45

**Solution** The standard atmospheric pressure is expressed in terms of mercury, water, and glycerin columns.

***Assumptions*** The densities of fluids are constant.

***Properties*** The specific gravities are given to be SG = 13.6 for mercury, SG = 1.0 for water, and SG = 1.26 for glycerin. The standard density of water is 1000 kg/m$^3$, and the standard atmospheric pressure is 101,325 Pa.

***Analysis*** The atmospheric pressure is expressed in terms of a fluid column height as

$$P_{atm} = \rho gh = SG\rho_w gh \quad \rightarrow \quad h = \frac{P_{atm}}{SG\rho_w g}$$

Substituting,

(*a*) Mercury: $$h = \frac{P_{atm}}{SG\rho_w g} = \frac{101{,}325 \text{ N/m}^2}{13.6(1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)}\left(\frac{1 \text{ kg} \cdot \text{m/s}^2}{1 \text{ N/m}^2}\right) = \mathbf{0.759 \ m}$$

(*b*) Water: $$h = \frac{P_{atm}}{SG\rho_w g} = \frac{101{,}325 \text{ N/m}^2}{1(1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)}\left(\frac{1 \text{ kg} \cdot \text{m/s}^2}{1 \text{ N/m}^2}\right) = \mathbf{10.3 \ m}$$

(*c*) Glycerin: $$h = \frac{P_{atm}}{SG\rho_w g} = \frac{101{,}325 \text{ N/m}^2}{1.26(1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)}\left(\frac{1 \text{ kg} \cdot \text{m/s}^2}{1 \text{ N/m}^2}\right) = \mathbf{8.20 \ m}$$

***Discussion*** Using water or glycerin to measure atmospheric pressure requires very long vertical tubes (over 10 m for water), which is not practical. This explains why mercury is used instead of water or a light fluid.

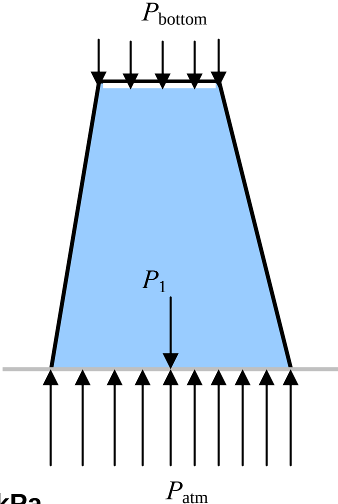
---

**3-46**

**Solution** A glass filled with water and covered with a thin paper is inverted. The pressure at the bottom of the glass is to be determined.

***Assumptions*** **1** Water is an incompressible substance. **2** The weight of the paper is negligible. **3** The atmospheric pressure is 100 kPa.

***Properties*** We take the density of water to be $\rho$ =1000 kg/m$^3$.




***Analysis*** The paper is in equilibrium, and thus the net force acting on the paper must be zero. A vertical force balance on the paper involves the pressure forces on both sides, and yields

$$P_1 A_{glass} = P_{atm} A_{glass} \quad \rightarrow \quad P_1 = P_{atm}$$

That is, the pressures on both sides of the paper must be the same.
The pressure at the bottom of the glass is determined from the hydrostatic pressure relation to be

$$P_{atm} = P_{bottom} + \rho g h_{glass} \quad \rightarrow \quad P_{bottom} = P_{atm} - \rho g h_{glass}$$

Substituting,

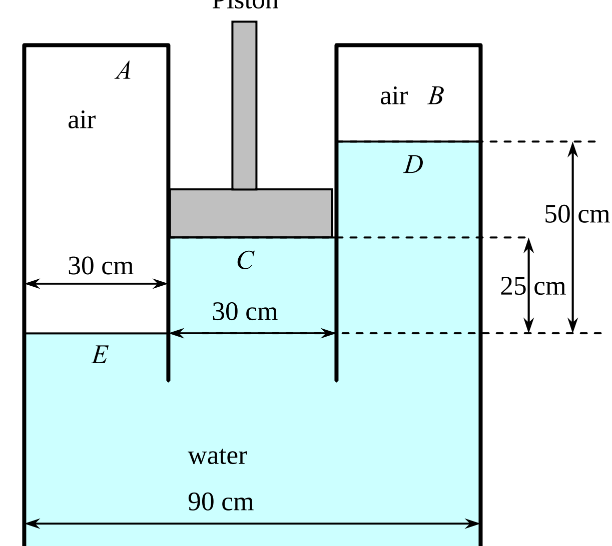
$$P_{bottom} = (100\text{ kPa}) - (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.1\text{ m})\left(\frac{1\text{N}}{1\text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{kPa}}{1000\text{ N/m}^2}\right) = \mathbf{99.0\ kPa}$$

***Discussion*** Note that there is a vacuum of 1 kPa at the bottom of the glass, and thus there is an upward pressure force acting on the water body, which balanced by the weight of water. As a result, the net downward force on water is zero, and thus water does not flow down.

---

**3-47**

**Solution** Two chambers with the same fluid at their base are separated by a piston. The gage pressure in each air chamber is to be determined.




***Assumptions*** **1** Water is an incompressible substance. **2** The variation of pressure with elevation in each air chamber is negligible because of the low density of air.

***Properties*** We take the density of water to be $\rho$ =1000 kg/m$^3$.

***Analysis*** The piston is in equilibrium, and thus the net force acting on the piston must be zero. A vertical force balance on the piston involves the pressure force exerted by water on the piston face, the atmospheric pressure force, and the piston weight, and yields

$$P_C A_{\text{piston}} = P_{\text{atm}} A_{\text{piston}} + W_{\text{piston}} \quad \rightarrow \quad P_C = P_{\text{atm}} + \frac{W_{\text{piston}}}{A_{\text{piston}}}$$

The pressure at the bottom of each air chamber is determined from the hydrostatic pressure relation to be

$$P_{\text{air A}} = P_E = P_C + \rho g \overline{CE} = P_{\text{atm}} + \frac{W_{\text{piston}}}{A_{\text{piston}}} + \rho g \overline{CE} \quad \rightarrow \quad P_{\text{air A, gage}} = \frac{W_{\text{piston}}}{A_{\text{piston}}} + \rho g \overline{CE}$$

$$P_{\text{air B}} = P_D = P_C - \rho g \overline{CD} = P_{\text{atm}} + \frac{W_{\text{piston}}}{A_{\text{piston}}} - \rho g \overline{CD} \quad \rightarrow \quad P_{\text{air B, gage}} = \frac{W_{\text{piston}}}{A_{\text{piston}}} - \rho g \overline{CD}$$

Substituting,

$$P_{\text{air A, gage}} = \frac{25\text{ N}}{\pi(0.3\text{ m})^2/4} + (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.25\text{ m})\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 2806\text{ N/m}^2 = \mathbf{2.81\ kPa}$$

$$P_{\text{air B, gage}} = \frac{25\text{ N}}{\pi(0.3\text{ m})^2/4} - (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.25\text{ m})\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = -2099\text{ N/m}^2 = \mathbf{-2.10\ kPa}$$

***Discussion*** Note that there is a vacuum of about 2 kPa in tank B which pulls the water up.

**3-48**

**Solution** A double-fluid manometer attached to an air pipe is considered. The specific gravity of one fluid is known, and the specific gravity of the other fluid is to be determined.

***Assumptions*** **1** Densities of liquids are constant. **2** The air pressure in the tank is uniform (i.e., its variation with elevation is negligible due to its low density), and thus the pressure at the air-water interface is the same as the indicated gage pressure.

***Properties*** The specific gravity of one fluid is given to be 13.55. We take the standard density of water to be 1000 kg/m$^3$.

***Analysis*** Starting with the pressure of air in the tank, and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the free surface where the oil tube is exposed to the atmosphere, and setting the result equal to $P_{atm}$ give

$$P_{air} + \rho_1 g h_1 - \rho_2 g h_2 = P_{atm} \quad \rightarrow \quad P_{air} - P_{atm} = SG_2 \rho_w g h_2 - SG_1 \rho_w g h_1$$

Rearranging and solving for $SG_2$,

$$SG_2 = SG_1 \frac{h_1}{h_2} + \frac{P_{air} - P_{atm}}{\rho_w g h_2} = 13.55 \frac{0.22 \text{ m}}{0.40 \text{ m}} + \left( \frac{(76 - 100) \text{ kPa}}{(1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(0.40 \text{ m})} \right) \left( \frac{1000 \text{ kg} \cdot \text{m/s}^2}{1 \text{ kPa} \cdot \text{m}^2} \right) = \mathbf{1.34}$$



***Discussion*** Note that the right fluid column is higher than the left, and this would imply above atmospheric pressure in the pipe for a single-fluid manometer.

**3-49**

**Solution** The pressure difference between two pipes is measured by a double-fluid manometer. For given fluid heights and specific gravities, the pressure difference between the pipes is to be calculated.

***Assumptions*** All the liquids are incompressible.

***Properties*** The specific gravities are given to be 13.5 for mercury, 1.26 for glycerin, and 0.88 for oil. We take the standard density of water to be $\rho_w$ =1000 kg/m$^3$.

***Analysis*** Starting with the pressure in the water pipe (point A) and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the oil pipe (point B), and setting the result equal to $P_B$ give

$$P_A + \rho_{\rm w} gh_w + \rho_{\rm Hg} gh_{\rm Hg} - \rho_{\rm gly} gh_{\rm gly} + \rho_{\rm pil} gh_{\rm oil} = P_B$$

Rearranging and using the definition of specific gravity,

$$P_B - P_A = {\rm SG}_w \rho_w gh_w + {\rm SG}_{Hg} \rho_w gh_{\rm Hg} - {\rm SG}_{gly} \rho_w gh_{\rm gly} + {\rm SG}_{oil} \rho_w gh_{\rm oil}$$

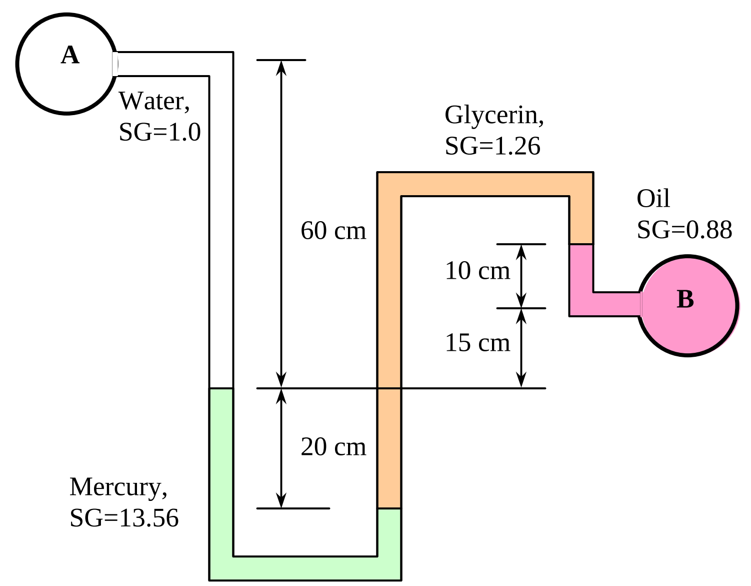
$$= g\rho_w \left({\rm SG}_w h_w + {\rm SG}_{Hg} h_{\rm Hg} - {\rm SG}_{gly} h_{\rm gly} + {\rm SG}_{oil} h_{\rm oil}\right)$$

Substituting,

$$P_B - P_A = (9.81\,{\rm m/s^2})(1000\,{\rm kg/m^3})[1(0.6\,{\rm m}) + 13.5(0.2\,{\rm m}) - 1.26(0.45\,{\rm m}) + 0.88(0.1\,{\rm m})]\left(\frac{1\,{\rm kN}}{1000\,{\rm kg\cdot m/s^2}}\right)$$

$$= 27.7\,{\rm kN/m^2} = \mathbf{27.7\,kPa}$$

Therefore, the pressure in the oil pipe is 27.7 kPa higher than the pressure in the water pipe.




***Discussion*** Using a manometer between two pipes is not recommended unless the pressures in the two pipes are relatively constant. Otherwise, an over-rise of pressure in one pipe can push the manometer fluid into the other pipe, creating a short circuit.

**3-50**

**Solution** The fluid levels in a multi-fluid U-tube manometer change as a result of a pressure drop in the trapped air space. For a given pressure drop and brine level change, the area ratio is to be determined.

***Assumptions*** **1** All the liquids are incompressible. **2** Pressure in the brine pipe remains constant. **3** The variation of pressure in the trapped air space is negligible.

***Properties*** The specific gravities are given to be 13.56 for mercury and 1.1 for brine. We take the standard density of water to be $\rho_w$ =1000 kg/m$^3$.

***Analysis*** It is clear from the problem statement and the figure that the brine pressure is much higher than the air pressure, and when the air pressure drops by 0.7 kPa, the pressure difference between the brine and the air space also increases by the same amount. Starting with the air pressure (point A) and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the brine pipe (point B), and setting the result equal to $P_B$ before and after the pressure change of air give

*Before*: $P_{A1} + \rho_w gh_w + \rho_{Hg} gh_{Hg,1} - \rho_{br} gh_{br,1} = P_B$

*After*: $P_{A2} + \rho_w gh_w + \rho_{Hg} gh_{Hg,2} - \rho_{br} gh_{br,2} = P_B$

Subtracting,

$$P_{A2} - P_{A1} + \rho_{Hg} g\Delta h_{Hg} - \rho_{br} g\Delta h_{br} = 0 \quad \rightarrow \quad \frac{P_{A1} - P_{A2}}{\rho_w g} = SG_{Hg}\Delta h_{Hg} - SG_{br}\Delta h_{br} = 0 \qquad (1)$$

where $\Delta h_{Hg}$ and $\Delta h_{br}$ are the changes in the differential mercury and brine column heights, respectively, due to the drop in air pressure. Both of these are positive quantities since as the mercury-brine interface drops, the differential fluid heights for both mercury and brine increase. Noting also that the volume of mercury is constant, we have $A_1\Delta h_{Hg,left} = A_2\Delta h_{Hg,right}$ and

$$P_{A2} - P_{A1} = -0.7 \text{ kPa} = -700 \text{ N/m}^2 = -700 \text{ kg/m} \cdot \text{s}^2$$

$$\Delta h_{br} = 0.005 \text{ m}$$

$$\Delta h_{Hg} = \Delta h_{Hg,right} + \Delta h_{Hg,left} = \Delta h_{br} + \Delta h_{br} A_2/\text{A}_1 = \Delta h_{br}(1 + A_2/\text{A}_1)$$

Substituting,

$$\frac{700 \text{ kg/m} \cdot \text{s}^2}{(1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)} = [13.56 \times 0.005(1 + A_2/A_1) - .1.1 \times 0.005] \text{ m}$$

It gives

$A_2/A_1$ = **0.134**

A
Air
Water
Area, $A_1$
Mercury
SG=13.56
B
Brine
pipe
SG=1.1
$\Delta h_b$ = 5 mm
Area, $A_2$

***Discussion*** In addition to the equations of hydrostatics, we also utilize conservation of mass in this problem.

**3-51**

**Solution** Two water tanks are connected to each other through a mercury manometer with inclined tubes. For a given pressure difference between the two tanks, the parameters $a$ and $\theta$ are to be determined.

***Assumptions*** Both water and mercury are incompressible liquids.

***Properties*** The specific gravity of mercury is given to be 13.6. We take the standard density of water to be $\rho_w$ =1000 kg/m$^3$.

***Analysis*** Starting with the pressure in the tank A and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach tank B, and setting the result equal to $P_B$ give

$$P_A + \rho_w ga + \rho_{Hg} g2a - \rho_w ga = P_B \quad \rightarrow \quad 2\rho_{Hg} ga = P_B - P_A$$

Rearranging and substituting the known values,

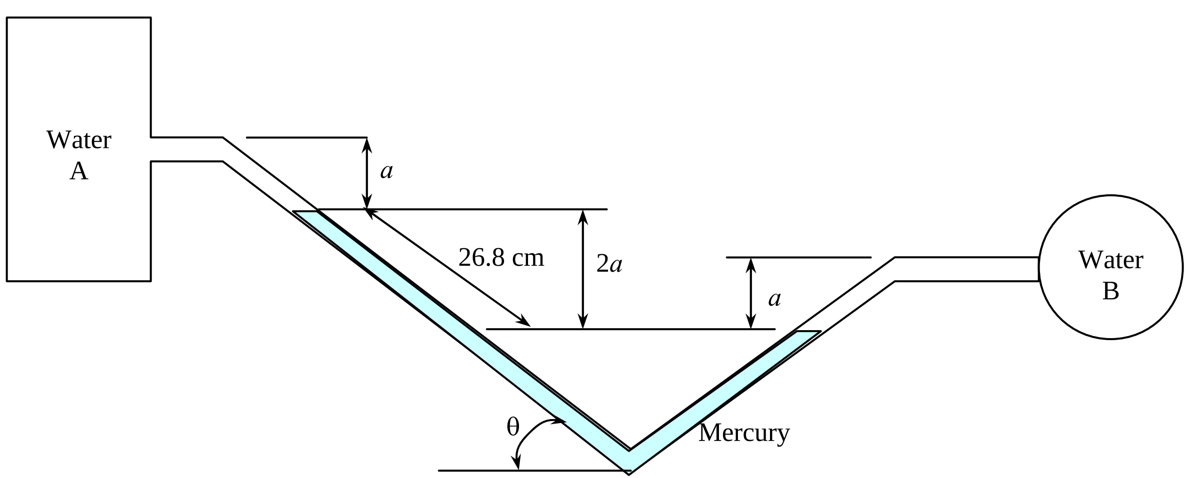
$$a = \frac{P_B - P_A}{2\rho_{Hg} g} = \frac{20 \text{ kN/m}^2}{2(13.6)(1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)}\left(\frac{1000 \text{ kg}\cdot\text{m/s}^2}{1 \text{ kN}}\right) = 0.0750 \text{ m} = \mathbf{7.50 \text{ cm}}$$

From geometric considerations,

$$26.8 \sin\theta = 2a \quad \text{(cm)}$$

Therefore,

$$\sin\theta = \frac{2a}{26.8} = \frac{2\times 7.50}{26.8} = 0.560 \qquad \rightarrow \qquad \theta = \mathbf{34.0°}$$




***Discussion*** Note that vertical distances are used in manometer analysis. Horizontal distances are of no consequence.

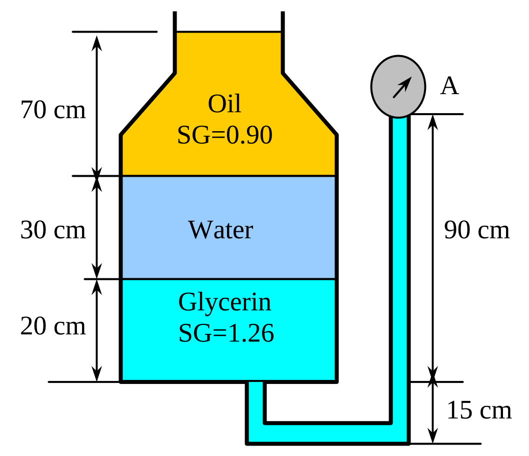
**3-52**

**Solution** A multi-fluid container is connected to a U-tube. For the given specific gravities and fluid column heights, the gage pressure at A and the height of a mercury column that would create the same pressure at A are to be determined.

***Assumptions*** **1** All the liquids are incompressible. **2** The multi-fluid container is open to the atmosphere.

***Properties*** The specific gravities are given to be 1.26 for glycerin and 0.90 for oil. We take the standard density of water to be $\rho_w$ =1000 kg/m$^3$, and the specific gravity of mercury to be 13.6.

***Analysis*** Starting with the atmospheric pressure on the top surface of the container and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach point $A$, and setting the result equal to $P_A$ give




$$P_{atm} + \rho_{\text{oil}} g h_{\text{oil}} + \rho_{\text{w}} g h_w - \rho_{\text{gly}} g h_{\text{gly}} = P_A$$

Rearranging and using the definition of specific gravity,

$$P_A - P_{atm} = \text{SG}_{oil}\rho_w g h_{\text{oil}} + \text{SG}_w \rho_w g h_w - \text{SG}_{gly}\rho_w g h_{\text{gly}}$$

or

$$P_{A,\text{gage}} = g\rho_w\left(\text{SG}_{\text{oil}}h_{\text{oil}} + \text{SG}_w h_w - \text{SG}_{\text{gly}}h_{\text{gly}}\right)$$

Substituting,

$$P_{A,gage} = (9.81\,\text{m/s}^2)(1000\,\text{kg/m}^3)[0.90(0.70\,\text{m}) + 1(0.3\,\text{m}) - 1.26(0.70\,\text{m})]\left(\frac{1\,\text{kN}}{1000\,\text{kg}\cdot\text{m/s}^2}\right)$$

$$= 0.471\,\text{kN/m}^2 = \mathbf{0.471\,kPa}$$

The equivalent mercury column height is

$$h_{Hg} = \frac{P_{A,gage}}{\rho_{Hg}\,g} = \frac{0.471\,\text{kN/m}^2}{(13.6)(1000\,\text{kg/m}^3)(9.81\,\text{m/s}^2)}\left(\frac{1000\,\text{kg}\cdot\text{m/s}^2}{1\,\text{kN}}\right) = 0.00353\,\text{m} = \mathbf{0.353\,cm}$$

***Discussion*** Note that the high density of mercury makes it a very suitable fluid for measuring high pressures in manometers.

## Fluid Statics: Hydrostatic Forces on Plane and Curved Surfaces

### 3-53C

**Solution** We are to define resultant force and center of pressure.

***Analysis*** The *resultant hydrostatic force* acting on a submerged surface is the **resultant of the pressure forces acting on the surface**. The **point of application of this resultant force** is called the *center of pressure*.

***Discussion*** The center of pressure is generally not at the center of the body, due to hydrostatic pressure variation.

---

### 3-54C

**Solution** We are to examine a claim about hydrostatic force.

***Analysis*** **Yes**, because the magnitude of the resultant force acting on a plane surface of a completely submerged body in a homogeneous fluid is equal to the product of the pressure $P_C$ at the centroid of the surface and the area $A$ of the surface. The pressure at the centroid of the surface is $P_C = P_0 + \rho g h_C$ where $h_C$ is the vertical distance of the centroid from the free surface of the liquid.

***Discussion*** We have assumed that we also know the pressure at the liquid surface.

---

### 3-55C

**Solution** We are to consider the effect of plate rotation on the hydrostatic force on the plate surface.

***Analysis*** There will be **no change** on the hydrostatic force acting on the top surface of this submerged horizontal flat plate as a result of this rotation since the magnitude of the resultant force acting on a plane surface of a completely submerged body in a homogeneous fluid is equal to the product of the pressure $P_C$ at the centroid of the surface and the area $A$ of the surface.

***Discussion*** If the rotation were not around the centroid, there *would* be a change in the force.

---

### 3-56C

**Solution** We are to explain why dams are bigger at the bottom than at the top.

***Analysis*** Dams are built much thicker at the bottom because **the pressure force increases with depth, and the bottom part of dams are subjected to largest forces**.

***Discussion*** Dam construction requires an enormous amount of concrete, so tapering the dam in this way saves a lot of concrete, and therefore a lot of money.

---

### 3-57C

**Solution** We are to explain how to determine the horizontal component of hydrostatic force on a curved surface.

***Analysis*** The horizontal component of the hydrostatic force acting on a curved surface is equal (in both magnitude and the line of action) to **the hydrostatic force acting on the vertical projection of the curved surface**.

***Discussion*** We could also integrate pressure along the surface, but the method discussed here is much simpler and yields the same answer.

---

**3-58C**

**Solution** We are to explain how to determine the vertical component of hydrostatic force on a curved surface.

***Analysis*** The vertical component of the hydrostatic force acting on a curved surface is equal to **the hydrostatic force acting on the horizontal projection of the curved surface**, **plus** (minus, if acting in the opposite direction) **the weight of the fluid block**.

***Discussion*** We could also integrate pressure along the surface, but the method discussed here is much simpler and yields the same answer.

---

**3-59C**

**Solution** We are to explain how to determine the line of action on a circular surface.

***Analysis*** The resultant hydrostatic force acting on a circular surface always passes through **the center of the circle** since the pressure forces are normal to the surface, and all lines normal to the surface of a circle pass through the center of the circle. Thus the pressure forces form a concurrent force system at the center, which can be reduced to a single equivalent force at that point. If the magnitudes of the horizontal and vertical components of the resultant hydrostatic force are known, the tangent of the angle the resultant hydrostatic force makes with the horizontal is $\boxed{\tan\alpha = F_V / F_H}$.

***Discussion*** This fact makes analysis of circular-shaped surfaces simple. There is no corresponding simplification for shapes other than circular, unfortunately.

---

**3-60**

**Solution** A car is submerged in water. The hydrostatic force on the door and its line of action are to be determined for the cases of the car containing atmospheric air and the car is filled with water.

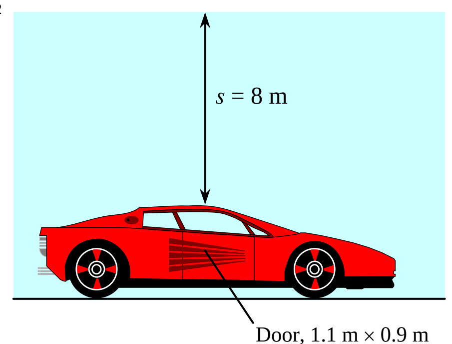
***Assumptions*** **1** The bottom surface of the lake is horizontal. **2** The door can be approximated as a vertical rectangular plate. **3** The pressure in the car remains at atmospheric value since there is no water leaking in, and thus no compression of the air inside. Therefore, we can ignore the atmospheric pressure in calculations since it acts on both sides of the door.

***Properties*** We take the density of lake water to be 1000 kg/m$^3$ throughout.

***Analysis*** (*a*) When the car is well-sealed and thus the pressure inside the car is the atmospheric pressure, the average pressure on the outer surface of the door is the pressure at the centroid (midpoint) of the surface, and is determined to be

$$P_{\text{avg}} = P_C = \rho g h_C = \rho g\left(s + b/2\right)$$

$$= \left(1000\ \text{kg/m}^3\right)\left(9.81\ \text{m/s}^2\right)\left(8 + 1.1/2\ \text{m}\right)\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) = 83.88\ \text{kN/m}^2$$




Then the resultant hydrostatic force on the door becomes

$$F_R = P_{ave}A = (83.88\ \text{kN/m}^2)(0.9\ \text{m} \times 1.1\ \text{m}) = \mathbf{83.0\ kN}$$

The pressure center is directly under the midpoint of the plate, and its distance from the surface of the lake is determined to be

$$y_P = s + \frac{b}{2} + \frac{b^2}{12(s+b/2)} = 8 + \frac{1.1}{2} + \frac{1.1^2}{12(8+1.1/2)} = \mathbf{8.56\ m}$$

(*b*) When the car is filled with water, the net force normal to the surface of the door is **zero** since the pressure on both sides of the door will be the same.

***Discussion*** Note that it is impossible for a person to open the door of the car when it is filled with atmospheric air. But it takes little effort to open the door when car is filled with water, because then the pressure on each side of the door is the same.

---

**3-61E**

**Solution** The height of a water reservoir is controlled by a cylindrical gate hinged to the reservoir. The hydrostatic force on the cylinder and the weight of the cylinder per ft length are to be determined.

***Assumptions*** **1** The hinge is frictionless. **2** Atmospheric pressure acts on both sides of the gate, and thus it can be ignored in calculations for convenience.

***Properties*** We take the density of water to be 62.4 lbm/ft$^3$ throughout.

***Analysis*** (*a*) We consider the free body diagram of the liquid block enclosed by the circular surface of the cylinder and its vertical and horizontal projections. The hydrostatic forces acting on the vertical and horizontal plane surfaces as well as the weight of the liquid block per ft length of the cylinder are:

Horizontal force on vertical surface:

$$F_H = F_x = P_{ave}A = \rho g h_C A = \rho g(s + R/2)A$$

$$= (62.4\text{ lbm/ft}^3)(32.2\text{ ft/s}^2)(13 + 2/2\text{ ft})(2\text{ ft} \times 1\text{ ft})\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right)$$

$$= 1747\text{ lbf}$$

Vertical force on horizontal surface (*upward*):

$$F_y = P_{\text{avg}}A = \rho g h_C A = \rho g h_{\text{bottom}} A$$

$$= \left(62.4\text{ lbm/ft}^3\right)\left(32.2\text{ ft/s}^2\right)\left(15\text{ ft}\right)\left(2\text{ ft} \times 1\text{ ft}\right)\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right)$$

$$= 1872\text{ lbf}$$

Weight of fluid block per ft length *(downward):*



$$W = mg = \rho g V = \rho g(R^2 - \pi R^2/4)(1\text{ ft}) = \rho g R^2(1 - \pi/4)(1\text{ ft})$$

$$= (62.4\text{ lbm/ft}^3)(32.2\text{ ft/s}^2)(2\text{ ft})^2(1\text{-}\pi/4)(1\text{ ft})\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right)$$

$$= 54\text{ lbf}$$

Therefore, the net upward vertical force is

$$F_V = F_y - W = 1872 - 54 = 1818\text{ lbf}$$

Then the magnitude and direction of the hydrostatic force acting on the cylindrical surface become

$$F_R = \sqrt{F_H^2 + F_V^2} = \sqrt{1747^2 + 1818^2} = 2521\text{ lbf} \cong \mathbf{2520\ lbf}$$

$$\tan\theta = \frac{F_V}{F_H} = \frac{1818\text{ lbf}}{1747\text{ lbf}} = 1.041 \quad\to\quad \theta = 46.1°$$

Therefore, the magnitude of the hydrostatic force acting on the cylinder is 2521 lbf per ft length of the cylinder, and its line of action passes through the center of the cylinder making an angle 46.1° upwards from the horizontal.

(*b*) When the water level is 15-ft high, the gate opens and the reaction force at the bottom of the cylinder becomes zero. Then the forces other than those at the hinge acting on the cylinder are its weight, acting through the center, and the hydrostatic force exerted by water. Taking a moment about the point *A* where the hinge is and equating it to zero gives

$$F_R R\sin\theta - W_{cyl}R = 0 \quad\to\quad W_{cyl} = F_R\sin\theta = (2521\ \text{lbf})\sin 46.1° = 1817\text{ lbf} \cong \mathbf{1820\ lbf}\ \ \text{(per ft)}$$

***Discussion*** The weight of the cylinder per ft length is determined to be 1820 lbf, which corresponds to a mass of 1820 lbm, and to a density of 145 lbm/ft$^3$ for the material of the cylinder.

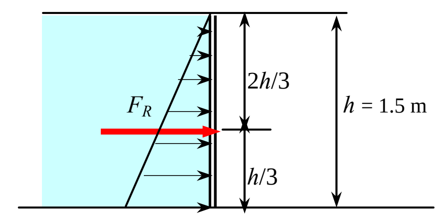
**3-62**

**Solution** An above the ground swimming pool is filled with water. The hydrostatic force on each wall and the distance of the line of action from the ground are to be determined, and the effect of doubling the wall height on the hydrostatic force is to be assessed.

***Assumptions*** Atmospheric pressure acts on both sides of the wall of the pool, and thus it can be ignored in calculations for convenience.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.

***Analysis*** The average pressure on a surface is the pressure at the centroid (midpoint) of the surface, and is determined to be




$$P_{avg} = P_C = \rho g h_C = \rho g(h/2)$$

$$= \left(1000 \text{ kg/m}^3\right)\left(9.81 \text{ m/s}^2\right)\left(1.5/2 \text{ m}\right)\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)$$

$$= 7357.5 \text{ N/m}^2$$

Then the resultant hydrostatic force on each wall becomes

$$F_R = P_{avg} A = \left(7357.5 \text{ N/m}^2\right)\left(4 \text{ m} \times 1.5 \text{ m}\right) = 44{,}145 \text{ N} \cong \mathbf{44.1\ kN}$$

The line of action of the force passes through the pressure center, which is 2*h*/3 from the free surface and *h*/3 from the bottom of the pool. Therefore, the distance of the line of action from the ground is

$$y_P = \frac{h}{3} = \frac{1.5}{3} = \mathbf{0.50\ m} \quad \text{(from the bottom)}$$

If the height of the walls of the pool is doubled, the hydrostatic force **quadruples** since

$$F_R = \rho g h_C A = \rho g (h/2)(h \times w) = \rho g w h^2/2$$

and thus the hydrostatic force is proportional to the square of the wall height, $h^2$.

***Discussion*** This is one reason why above-ground swimming pools are not very deep, whereas in-ground swimming pools can be quite deep.

**3-63E**

**Solution** A dam is filled to capacity. The total hydrostatic force on the dam, and the pressures at the top and the bottom are to be determined.

***Assumptions*** Atmospheric pressure acts on both sides of the dam, and thus it can be ignored in calculations for convenience.

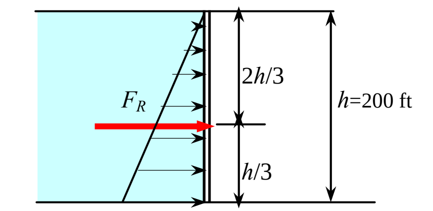
***Properties*** We take the density of water to be 62.4 lbm/ft$^3$ throughout.

***Analysis*** The average pressure on a surface is the pressure at the centroid (midpoint) of the surface, and is determined to be

$$P_{\text{avg}} = \rho g h_C = \rho g (h/2)$$

$$= (62.4 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)(200/2 \text{ ft})\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm}\cdot\text{ft/s}^2}\right)$$

$$= 6240 \text{ lbf/ft}^2$$




Then the resultant hydrostatic force acting on the dam becomes

$$F_R = P_{\text{ave}} A = (6240 \text{ lbf/ft}^2)(200 \text{ ft} \times 1200 \text{ ft}) = \mathbf{1.50 \times 10^9 \text{ lbf}}$$

Resultant force per unit area is pressure, and its value at the top and the bottom of the dam becomes

$$P_{\text{top}} = \rho g h_{\text{top}} = \mathbf{0 \text{ lbf/ft}^2}$$

$$P_{\text{bottom}} = \rho g h_{\text{bottom}} = (62.4 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)(200 \text{ ft})\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm}\cdot\text{ft/s}^2}\right) = 12{,}480 \text{ lbf/ft}^2 \cong \mathbf{12{,}500 \text{ lbf/ft}^2}$$

***Discussion*** The values above are gave pressures, of course. The gage pressure at the bottom of the dam is about 86.6 psig, or 101.4 psia, which is almost seven times greater than standard atmospheric pressure.

---

**3-64**

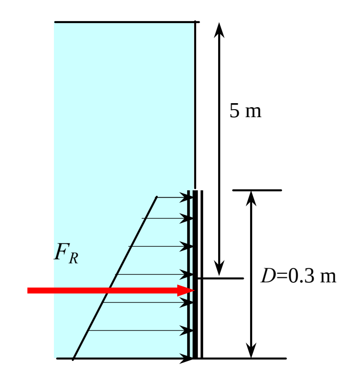
**Solution** A room in the lower level of a cruise ship is considered. The hydrostatic force acting on the window and the pressure center are to be determined.

***Assumptions*** Atmospheric pressure acts on both sides of the window, and thus it can be ignored in calculations for convenience.

***Properties*** The specific gravity of sea water is given to be 1.025, and thus its density is 1025 kg/m$^3$.

***Analysis*** The average pressure on a surface is the pressure at the centroid (midpoint) of the surface, and is determined to be

$$P_{avg} = P_C = \rho g h_C = (1025 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(5 \text{ m})\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = 50{,}276 \text{ N/m}^2$$




Then the resultant hydrostatic force on each wall becomes

$$F_R = P_{\text{avg}} A = P_{\text{avg}}[\pi D^2/4] = (50{,}276 \text{ N/m}^2)[\pi(0.3 \text{ m})^2/4] = 3554 \text{ N} \cong \mathbf{3550 \text{ N}}$$

The line of action of the force passes through the pressure center, whose vertical distance from the free surface is determined from

$$y_P = y_C + \frac{I_{xx,C}}{y_C A} = y_C + \frac{\pi R^4/4}{y_C \pi R^2} = y_C + \frac{R^2}{4y_C} = 5 + \frac{(0.15 \text{ m})^2}{4(5 \text{ m})} = 5.0011 \text{ m} \cong \mathbf{5.00 \text{ m}}$$

***Discussion*** For small surfaces deep in a liquid, the pressure center nearly coincides with the centroid of the surface. Here, in fact, to three significant digits in the final answer, the center of pressure and centroid are coincident.

---

**3-65**

**Solution** The cross-section of a dam is a quarter-circle. The hydrostatic force on the dam and its line of action are to be determined.

***Assumptions*** Atmospheric pressure acts on both sides of the dam, and thus it can be ignored in calculations for convenience.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.

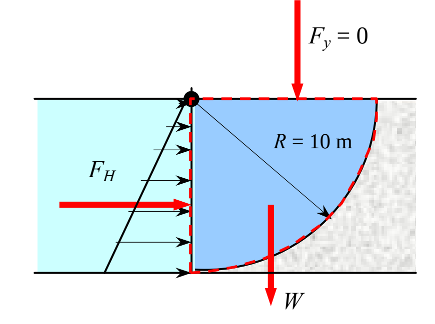
***Analysis*** We consider the free body diagram of the liquid block enclosed by the circular surface of the dam and its vertical and horizontal projections. The hydrostatic forces acting on the vertical and horizontal plane surfaces as well as the weight of the liquid block are:

Horizontal force on vertical surface:

$$F_H = F_x = P_{avg}A = \rho g h_C A = \rho g(R/2)A$$

$$= (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(10/2 \text{ m})(10 \text{ m} \times 100 \text{ m})\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)$$

$$= 4.905\times10^7 \text{ N}$$




Vertical force on horizontal surface is zero since it coincides with the free surface of water. The weight of fluid block per m length is

$$F_V = W = \rho g V = \rho g[w\times \pi R^2/4]$$

$$= (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)[(100 \text{ m})\pi(10 \text{ m})^2/4]\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)$$

$$= 7.705\times10^7 \text{ N}$$

Then the magnitude and direction of the hydrostatic force acting on the surface of the dam become

$$F_R = \sqrt{F_H^2 + F_V^2} = \sqrt{(4.905\times10^7 \text{ N})^2 + (7.705\times10^7 \text{ N})^2} = 9.134\times10^7 \text{ N} \cong \mathbf{9.13\times10^7\ N}$$

$$\tan\theta = \frac{F_V}{F_H} = \frac{7.705\times10^7 \text{ N}}{4.905\times10^7 \text{ N}} = 1.571 \quad \rightarrow \quad \theta = \mathbf{57.5°}$$

Therefore, the line of action of the hydrostatic force passes through the center of the curvature of the dam, making 57.5° downwards from the horizontal.

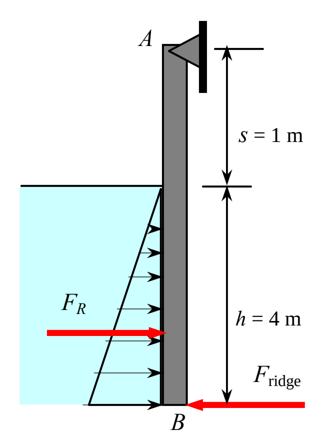
***Discussion*** If the shape were not circular, it would be more difficult to determine the line of action.

**3-66**

**Solution** A rectangular plate hinged about a horizontal axis along its upper edge blocks a fresh water channel. The plate is restrained from opening by a fixed ridge at a point *B*. The force exerted to the plate by the ridge is to be determined.

***Assumptions*** Atmospheric pressure acts on both sides of the plate, and thus it can be ignored in calculations for convenience.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.




***Analysis*** The average pressure on a surface is the pressure at the centroid (midpoint) of the surface, and is determined to be

$$P_{avg} = P_C = \rho g h_C = \rho g(h/2)$$

$$= (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(4/2 \text{ m})\left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right) = 19.62 \text{ kN/m}^2$$

Then the resultant hydrostatic force on each wall becomes

$$F_R = P_{avg} A = (19.62 \text{ kN/m}^2)(4 \text{ m} \times 5 \text{ m}) = 392 \text{ kN}$$

The line of action of the force passes through the pressure center, which is 2*h*/3 from the free surface,

$$y_P = \frac{2h}{3} = \frac{2 \times (4 \text{ m})}{3} = 2.667 \text{ m}$$

Taking the moment about point *A* and setting it equal to zero gives

$$\sum M_A = 0 \quad \rightarrow \quad F_R(s + y_P) = F_{ridge}\overline{AB}$$

Solving for $F_{ridge}$ and substituting, the reaction force is determined to be

$$F_{ridge} = \frac{s + y_P}{\overline{AB}} F_R = \frac{(1 + 2.667) \text{ m}}{5 \text{ m}}(392 \text{ kN}) = \mathbf{288 \text{ kN}}$$

***Discussion*** The difference between $F_R$ and $F_{ridge}$ is the force acting on the hinge at point *A*.


**3-67** 

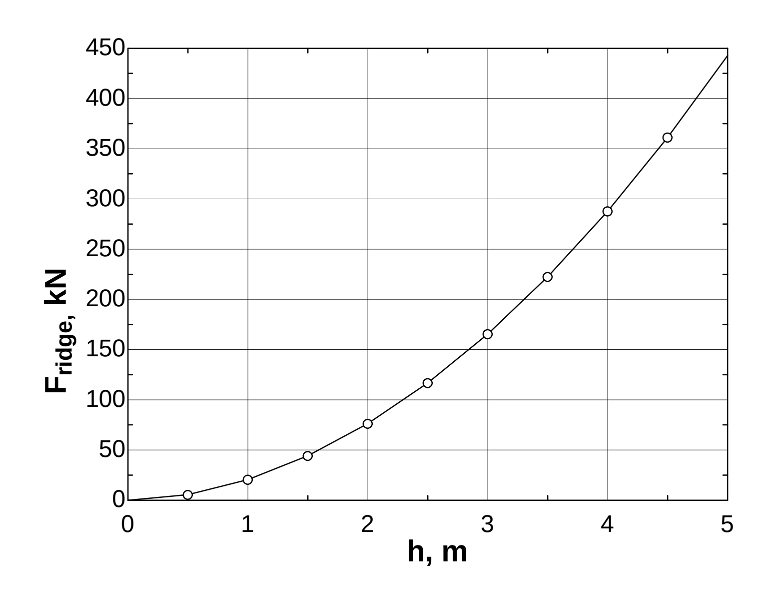
**Solution** The previous problem is reconsidered. The effect of water depth on the force exerted on the plate by the ridge as the water depth varies from 0 to 5 m in increments of 0.5 m is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
g=9.81 "m/s2"
rho=1000 "kg/m3"
s=1"m"

w=5 "m"
A=w*h
P_ave=rho*g*h/2000 "kPa"
F_R=P_ave*A "kN"
y_p=2*h/3
F_ridge=(s+y_p)*F_R/(s+h)
```

| Dept *h*, m | $P_{ave}$, kPa | $F_R$ kN | $y_p$ m | $F_{ridge}$ kN |
|---|---|---|---|---|
| 0.0 | 0 | 0.0 | 0.00 | 0 |
| 0.5 | 2.453 | 6.1 | 0.33 | 5 |
| 1.0 | 4.905 | 24.5 | 0.67 | 20 |
| 1.5 | 7.358 | 55.2 | 1.00 | 44 |
| 2.0 | 9.81 | 98.1 | 1.33 | 76 |
| 2.5 | 12.26 | 153.3 | 1.67 | 117 |
| 3.0 | 14.72 | 220.7 | 2.00 | 166 |
| 3.5 | 17.17 | 300.4 | 2.33 | 223 |
| 4.0 | 19.62 | 392.4 | 2.67 | 288 |
| 4.5 | 22.07 | 496.6 | 3.00 | 361 |
| 5.0 | 24.53 | 613.1 | 3.33 | 443 |



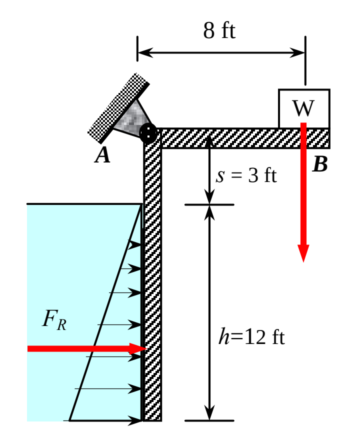

***Discussion*** The force on the ridge does not increase linearly, as we may have suspected.

**3-68E**

**Solution** The flow of water from a reservoir is controlled by an L-shaped gate hinged at a point *A*. The required weight *W* for the gate to open at a specified water height is to be determined.

***Assumptions*** **1** Atmospheric pressure acts on both sides of the gate, and thus it can be ignored in calculations for convenience. **2** The weight of the gate is negligible.

***Properties*** We take the density of water to be 62.4 lbm/ft$^3$ throughout.




***Analysis*** The average pressure on a surface is the pressure at the centroid (midpoint) of the surface, and is determined to be

$$P_{\text{avg}} = \rho g h_C = \rho g (h/2)$$

$$= (62.4\ \text{lbm/ft}^3)(32.2\ \text{ft/s}^2)(12/2\ \text{ft})\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right)$$

$$= 374.4\ \text{lbf/ft}^2$$



Then the resultant hydrostatic force acting on the dam becomes

$$F_R = P_{\text{avg}} A = (374.4\ \text{lbf/ft}^2)(12\ \text{ft} \times 5\ \text{ft}) = 22{,}464\ \text{lbf}$$

The line of action of the force passes through the pressure center, which is $2h/3$ from the free surface,

$$y_P = \frac{2h}{3} = \frac{2 \times (12\ \text{ft})}{3} = 8\ \text{ft}$$

Taking the moment about point *A* and setting it equal to zero gives

$$\sum M_A = 0 \quad \rightarrow \quad F_R (s + y_P) = W\overline{AB}$$

Solving for *W* and substituting, the required weight is determined to be

$$W = \frac{s + y_P}{\overline{AB}} F_R = \frac{(3+8)\ \text{ft}}{8\ \text{ft}} (22{,}464\ \text{lbf}) = \mathbf{30{,}900\ lbf}$$

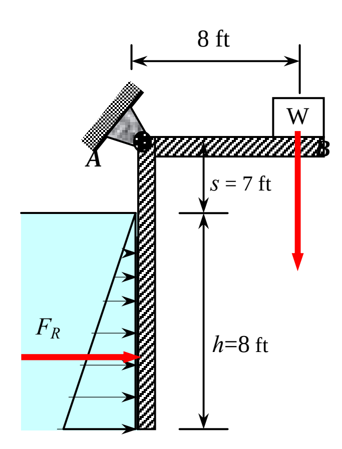
***Discussion*** Note that the required weight is inversely proportional to the distance of the weight from the hinge.

**3-69E**

**Solution** The flow of water from a reservoir is controlled by an L-shaped gate hinged at a point *A*. The required weight *W* for the gate to open at a specified water height is to be determined.

***Assumptions*** **1** Atmospheric pressure acts on both sides of the gate, and thus it can be ignored in calculations for convenience. **2** The weight of the gate is negligible.

***Properties*** We take the density of water to be 62.4 lbm/ft³ throughout.




***Analysis*** The average pressure on a surface is the pressure at the centroid (midpoint) of the surface, and is determined to be

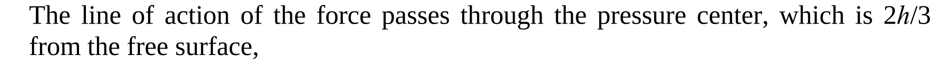


$$P_{avg} = \rho g h_C = \rho g (h/2)$$

$$= (62.4 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)(8/2 \text{ ft})\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm}\cdot\text{ft/s}^2}\right)$$

$$= 249.6 \text{ lbf/ft}^2$$

Then the resultant hydrostatic force acting on the dam becomes



$$F_R = P_{avg} A = (249.6 \text{ lbf/ft}^2)(8 \text{ ft} \times 5 \text{ ft}) = 9984 \text{ lbf}$$

The line of action of the force passes through the pressure center, which is $2h/3$ from the free surface,

$$y_P = \frac{2h}{3} = \frac{2\times(8 \text{ ft})}{3} = 5.333 \text{ ft}$$

Taking the moment about point *A* and setting it equal to zero gives

$$\sum M_A = 0 \quad \rightarrow \quad F_R(s + y_P) = W\overline{AB}$$

Solving for *W* and substituting, the required weight is determined to be

$$W = \frac{s + y_P}{\overline{AB}} F_R = \frac{(7 + 5.333) \text{ ft}}{8 \text{ ft}}(9984 \text{ lbf}) = 15,390 \text{ lbf} \cong \mathbf{15,400 \text{ lbf}}$$

***Discussion*** Note that the required weight is inversely proportional to the distance of the weight from the hinge.

**3-70**

**Solution** Two parts of a water trough of semi-circular cross-section are held together by cables placed along the length of the trough. The tension $T$ in each cable when the trough is full is to be determined.

***Assumptions*** **1** Atmospheric pressure acts on both sides of the trough wall, and thus it can be ignored in calculations for convenience. **2** The weight of the trough is negligible.

***Properties*** We take the density of water to be 1000 kg/m³ throughout.

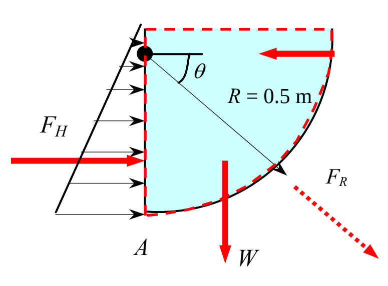
***Analysis*** To expose the cable tension, we consider half of the trough whose cross-section is quarter-circle. The hydrostatic forces acting on the vertical and horizontal plane surfaces as well as the weight of the liquid block are:

Horizontal force on vertical surface:

$$F_H = F_x = P_{\text{avg}}A = \rho g h_C A = \rho g(R/2)A$$

$$= \left(1000 \text{ kg/m}^3\right)\left(9.81 \text{ m/s}^2\right)\left(0.5/2 \text{ m}\right)(0.5 \text{ m} \times 3 \text{ m})\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)$$

$$= 3679 \text{ N}$$




The vertical force on the horizontal surface is zero, since it coincides with the free surface of water. The weight of fluid block per 3-m length is

$$F_V = W = \rho g V = \rho g[w \times \pi R^2/4]$$

$$= (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)[(3 \text{ m})\pi(0.5 \text{ m})^2/4]\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)$$

$$= 5779 \text{ N}$$

Then the magnitude and direction of the hydrostatic force acting on the surface of the 3-m long section of the trough become

$$F_R = \sqrt{F_H^2 + F_V^2} = \sqrt{(3679 \text{ N})^2 + (5779 \text{ N})^2} = 6851 \text{ N}$$

$$\tan\theta = \frac{F_V}{F_H} = \frac{5779 \text{ N}}{3679 \text{ N}} = 1.571 \quad \rightarrow \quad \theta = 57.5°$$

Therefore, the line of action passes through the center of the curvature of the trough, making 57.5° downwards from the horizontal. Taking the moment about point $A$ where the two parts are hinged and setting it equal to zero gives

$$\sum M_A = 0 \quad \rightarrow \quad F_R R\sin(90-57.5)° = TR$$

Solving for $T$ and substituting, the tension in the cable is determined to be

$$T = F_R\sin(90-57.5)° = (6851 \text{ N})\sin(90-57.5)° = 3681 \text{ N} \cong \mathbf{3680 \text{ N}}$$

***Discussion*** This problem can also be solved without finding $F_R$ by finding the lines of action of the horizontal hydrostatic force and the weight.

**3-71**

**Solution** Two parts of a water trough of triangular cross-section are held together by cables placed along the length of the trough. The tension $T$ in each cable when the trough is filled to the rim is to be determined.

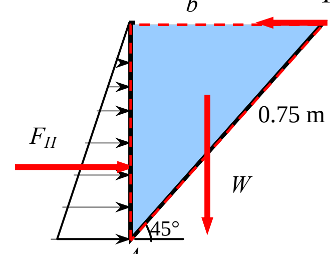
***Assumptions*** **1** Atmospheric pressure acts on both sides of the trough wall, and thus it can be ignored in calculations for convenience. **2** The weight of the trough is negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.

***Analysis*** To expose the cable tension, we consider half of the trough whose cross-section is triangular. The water height $h$ at the midsection of the trough and width of the free surface are

$$h = L\sin\theta = (0.75\text{ m})\sin 45° = 0.530\text{ m}$$

$$b = L\cos\theta = (0.75\text{ m})\cos 45° = 0.530\text{ m}$$




The hydrostatic forces acting on the vertical and horizontal plane surfaces as well as the weight of the liquid block are determined as follows:

Horizontal force on vertical surface:

$$F_H = F_x = P_{avg}A = \rho g h_C A = \rho g (h/2) A$$

$$= (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.530/2\text{ m})(0.530\text{ m} \times 6\text{ m})\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)$$

$$= 8267\text{ N}$$

The vertical force on the horizontal surface is zero since it coincides with the free surface of water. The weight of fluid block per 6-m length is

$$F_V = W = \rho g V = \rho g[w \times bh/2]$$

$$= (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)[(6\text{ m})(0.530\text{ m})(0.530\text{ m})/2]\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)$$

$$= 8267\text{ N}$$

The distance of the centroid of a triangle from a side is 1/3 of the height of the triangle for that side. Taking the moment about point $A$ where the two parts are hinged and setting it equal to zero gives

$$\sum M_A = 0 \quad \rightarrow \quad W\frac{b}{3} + F_H\frac{h}{3} = Th$$

Solving for $T$ and substituting, and noting that $h = b$, the tension in the cable is determined to be

$$T = \frac{F_H + W}{3} = \frac{(8267 + 8267)\text{ N}}{3} = 5511\text{ N} \cong \mathbf{5510\text{ N}}$$

***Discussion*** The analysis is simplified because of the symmetry of the trough.

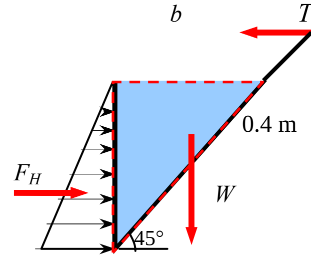
**3-72**

**Solution** Two parts of a water trough of triangular cross-section are held together by cables placed along the length of the trough. The tension $T$ in each cable when the trough is filled to the rim is to be determined.

***Assumptions*** **1** Atmospheric pressure acts on both sides of the trough wall, and thus it can be ignored in calculations for convenience. **2** The weight of the trough is negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.

***Analysis*** To expose the cable tension, we consider half of the trough whose cross-section is triangular. The water height is given to be $h$ = 0.4 m at the midsection of the trough, which is equivalent to the width of the free surface $b$ since tan 45° = $b/h$ = 1. The hydrostatic forces acting on the vertical and horizontal plane surfaces as well as the weight of the liquid block are determined as follows:




Horizontal force on vertical surface:

$$F_H = F_x = P_{avg}A = \rho g h_C A = \rho g\left(h/2\right)A$$
$$= \left(1000\ \text{kg/m}^3\right)\left(9.81\ \text{m/s}^2\right)\left(0.4/2\ \text{m}\right)(0.4\ \text{m}\ \times\ 3\ \text{m})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)$$
$$= 2354\ \text{N}$$

The vertical force on the horizontal surface is zero since it coincides with the free surface of water. The weight of fluid block per 3-m length is

$$F_V = W = \rho g V = \rho g[w \times bh/2]$$
$$= (1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)[(3\ \text{m})(0.4\ \text{m})(0.4\ \text{m})/2]\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)$$
$$= 2354\ \text{N}$$

The distance of the centroid of a triangle from a side is 1/3 of the height of the triangle for that side. Taking the moment about point $A$ where the two parts are hinged and setting it equal to zero gives

$$\sum M_A = 0 \qquad \rightarrow \qquad W\frac{b}{3} + F_H\frac{h}{3} = Th$$

Solving for $T$ and substituting, and noting that $h = b$, the tension in the cable is determined to be

$$T = \frac{F_H + W}{3} = \frac{\left(2354 + 2354\right)\ \text{N}}{3} = 1569\ \text{N} \cong \mathbf{1570\ N}$$

***Discussion*** The tension force here is a factor of about 3.5 smaller than that of the previous problem, even though the trough is more than half full.

**3-73**

**Solution** A retaining wall against mud slide is to be constructed by rectangular concrete blocks. The mud height at which the blocks will start sliding, and the blocks will tip over are to be determined.

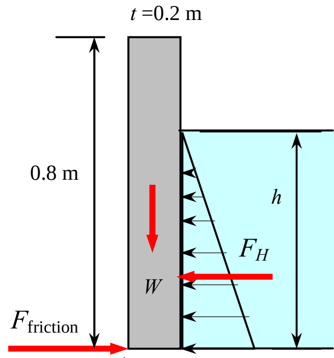
***Assumptions*** Atmospheric pressure acts on both sides of the wall, and thus it can be ignored in calculations for convenience.

***Properties*** The density is given to be 1800 kg/m$^3$ for the mud, and 2700 kg/m$^3$ for concrete blocks.

***Analysis*** (*a*) The weight of the concrete wall per unit length ($L$ = 1 m) and the friction force between the wall and the ground are

$$W_{\text{block}} = \rho g V = (2700\text{ kg/m}^3)(9.81\text{ m/s}^2)[0.2\times 0.8\times 1\text{ m}^3]\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 4238\text{ N}$$

$$F_{\text{friction}} = \mu W_{\text{block}} = 0.3(4238\text{ N}) = 1271\text{ N}$$




The hydrostatic force exerted by the mud to the wall is

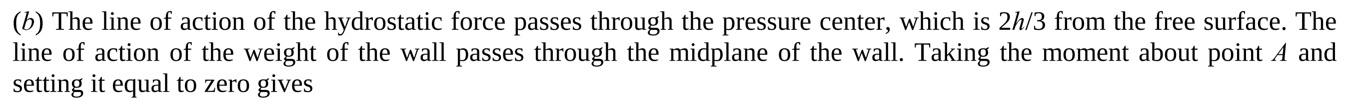
$$F_H = F_x = P_{\text{avg}}A = \rho g h_C A = \rho g (h/2) A$$

$$= (1800\text{ kg/m}^3)(9.81\text{ m/s}^2)(h/2)(1\times h)\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)$$

$$= 8829h^2\text{ N}$$

Setting the hydrostatic and friction forces equal to each other gives

$$F_H = F_{\text{friction}} \quad\rightarrow\quad 8829\,h^2 = 1271 \quad\rightarrow\quad h = \mathbf{0.38\ m}$$



(*b*) The line of action of the hydrostatic force passes through the pressure center, which is 2$h$/3 from the free surface. The line of action of the weight of the wall passes through the midplane of the wall. Taking the moment about point $A$ and setting it equal to zero gives

$$\sum M_A = 0 \quad\rightarrow\quad W_{\text{block}}(t/2) = F_H(h/3) \quad\rightarrow\quad W_{\text{block}}(t/2) = 8829h^3/3$$

Solving for $h$ and substituting, the mud height for tip over is determined to be

$$h = \left(\frac{3W_{\text{block}}t}{2\times 8829}\right)^{1/3} = \left(\frac{3\times 4238\times 0.2}{2\times 8829}\right)^{1/3} = \mathbf{0.52\ m}$$

***Discussion*** The concrete wall will slide before tipping. Therefore, sliding is more critical than tipping in this case.

**3-74**

**Solution** A retaining wall against mud slide is to be constructed by rectangular concrete blocks. The mud height at which the blocks will start sliding, and the blocks will tip over are to be determined.

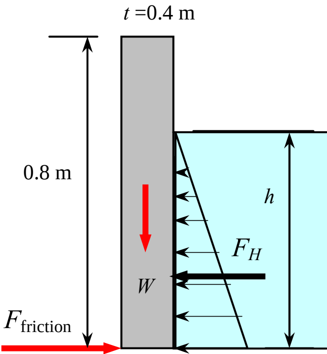
***Assumptions*** Atmospheric pressure acts on both sides of the wall, and thus it can be ignored in calculations for convenience.

***Properties*** The density is given to be 1800 kg/m$^3$ for the mud, and 2700 kg/m$^3$ for concrete blocks.

***Analysis*** (*a*) The weight of the concrete wall per unit length ($L$ = 1 m) and the friction force between the wall and the ground are

$$W_{\text{block}} = \rho g V = (2700\ \text{kg/m}^3)(9.81\ \text{m/s}^2)[0.4\times 0.8\times 1\ \text{m}^3]\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = 8476\ \text{N}$$

$$F_{\text{friction}} = \mu W_{\text{block}} = 0.3(8476\ \text{N}) = 2543\ \text{N}$$



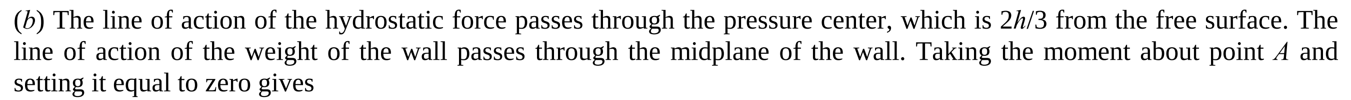

The hydrostatic force exerted by the mud to the wall is

$$F_H = F_x = P_{\text{avg}}A = \rho g h_C A = \rho g\left(h/2\right)A$$

$$= \left(1800\ \text{kg/m}^3\right)\left(9.81\ \text{m/s}^2\right)\left(h/2\right)(1\times h)\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)$$

$$= 8829h^2\ \text{N}$$

Setting the hydrostatic and friction forces equal to each other gives



$$F_H = F_{\text{friction}} \quad\rightarrow\quad 8829\,h^2 = 2543 \quad\rightarrow\quad h = \mathbf{0.54\ m}$$

(*b*) The line of action of the hydrostatic force passes through the pressure center, which is 2*h*/3 from the free surface. The line of action of the weight of the wall passes through the midplane of the wall. Taking the moment about point *A* and setting it equal to zero gives

$$\sum M_A = 0 \quad\rightarrow\quad W_{\text{block}}(t/2) = F_H(h/3) \quad\rightarrow\quad W_{\text{block}}(t/2) = 8829h^3/3$$

Solving for *h* and substituting, the mud height for tip over is determined to be

$$h = \left(\frac{3W_{\text{block}}t}{2\times 8829}\right)^{1/3} = \left(\frac{3\times 8476\times 0.3}{2\times 8829}\right)^{1/3} = \mathbf{0.76\ m}$$

***Discussion*** Note that the concrete wall will slide before tipping. Therefore, sliding is more critical than tipping in this case.

**3-75** **[*Also solved using EES on enclosed DVD*]**

**Solution** A quarter-circular gate hinged about its upper edge controls the flow of water over the ledge at *B* where the gate is pressed by a spring. The minimum spring force required to keep the gate closed when the water level rises to *A* at the upper edge of the gate is to be determined.

***Assumptions*** **1** The hinge is frictionless. **2** Atmospheric pressure acts on both sides of the gate, and thus it can be ignored in calculations for convenience. **3** The weight of the gate is negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.

***Analysis*** We consider the free body diagram of the liquid block enclosed by the circular surface of the gate and its vertical and horizontal projections. The hydrostatic forces acting on the vertical and horizontal plane surfaces as well as the weight of the liquid block are determined as follows:

A
$F_x$
W
R = 3 m
$F_s$
B
$F_y$

*Horizontal force on vertical surface*:

$$F_H = F_x = P_{ave}A = \rho g h_C A = \rho g(R/2)A$$

$$= (1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(3/2\ \text{m})(4\ \text{m}\times 3\ \text{m})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)$$

$$= 176.6\ \text{kN}$$

*Vertical force on horizontal surface* (upward):

$$F_y = P_{\text{avg}}A = \rho g h_C A = \rho g h_{\text{bottom}} A$$

$$= \left(1000\ \text{kg/m}^3\right)\left(9.81\ \text{m/s}^2\right)\left(3\ \text{m}\right)\left(4\ \text{m}\ \times\ 3\ \text{m}\right)\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) = 353.2\ \text{kN}$$

*The weight of fluid block per 4-m length* (downwards):

$$W = \rho g V = \rho g\left[w\times \pi R^2/4\right]$$

$$= \left(1000\ \text{kg/m}^3\right)\left(9.81\ \text{m/s}^2\right)\left[\left(4\ \text{m}\right)\pi(3\ \text{m})^2/4\right]\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) = 277.4\ \text{kN}$$

Therefore, the net upward vertical force is

$$F_V = F_y - W = 353.2 - 277.4 = 75.8\ \text{kN}$$

Then the magnitude and direction of the hydrostatic force acting on the surface of the 4-m long quarter-circular section of the gate become

$$F_R = \sqrt{F_H^2 + F_V^2} = \sqrt{(176.6\ \text{kN})^2 + (75.8\ \text{kN})^2} = 192.2\ \text{kN}$$

$$\tan\theta = \frac{F_V}{F_H} = \frac{75.8\ \text{kN}}{176.6\ \text{kN}} = 0.429 \quad \rightarrow \quad \theta = 23.2°$$

Therefore, the magnitude of the hydrostatic force acting on the gate is 192.2 kN, and its line of action passes through the center of the quarter-circular gate making an angle 23.2° upwards from the horizontal.

The minimum spring force needed is determined by taking a moment about the point *A* where the hinge is, and setting it equal to zero,

$$\sum M_A = 0 \quad \rightarrow \quad F_R R\sin(90-\theta) - F_{\text{spring}} R = 0$$

Solving for $F_{\text{spring}}$ and substituting, the spring force is determined to be

$$F_{\text{spring}} = F_R \sin(90\text{-}\theta) = (192.2\ \text{kN})\sin(90° - 23.2°) = \mathbf{177\ kN}$$

***Discussion*** Several variations of this design are possible. Can you think of some of them?

**3-76**

**Solution** A quarter-circular gate hinged about its upper edge controls the flow of water over the ledge at *B* where the gate is pressed by a spring. The minimum spring force required to keep the gate closed when the water level rises to *A* at the upper edge of the gate is to be determined.

***Assumptions*** **1** The hinge is frictionless. **2** Atmospheric pressure acts on both sides of the gate, and thus it can be ignored in calculations for convenience. **3** The weight of the gate is negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.

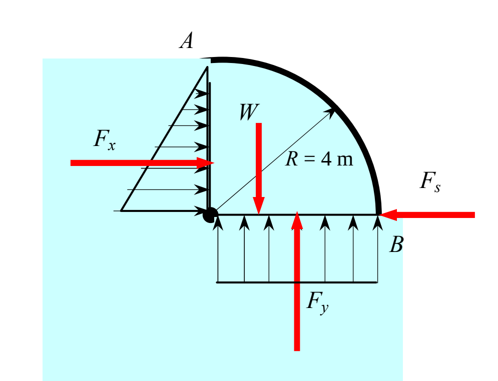
***Analysis*** We consider the free body diagram of the liquid block enclosed by the circular surface of the gate and its vertical and horizontal projections. The hydrostatic forces acting on the vertical and horizontal plane surfaces as well as the weight of the liquid block are determined as follows:

*Horizontal force on vertical surface*:

$$F_H = F_x = P_{ave}A = \rho g h_C A = \rho g(R/2)A$$

$$= (1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(4/2\ \text{m})(4\ \text{m}\times 4\ \text{m})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)$$

$$= 313.9\ \text{kN}$$




*Vertical force on horizontal surface* (upward):

$$F_y = P_{ave}A = \rho g h_C A = \rho g h_{\text{bottom}} A$$

$$= (1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(4\ \text{m})(4\ \text{m}\times 4\ \text{m})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)$$

$$= 627.8\ \text{kN}$$

*The weight of fluid block per 4-m length* (downwards):

$$W = \rho g V = \rho g[w\times \pi R^2/4]$$

$$= (1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)[(4\ \text{m})\pi(4\ \text{m})^2/4]\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)$$

$$= 493.1\ \text{kN}$$

Therefore, the net upward vertical force is

$$F_V = F_y - W = 627.8 - 493.1 = 134.7\ \text{kN}$$

Then the magnitude and direction of the hydrostatic force acting on the surface of the 4-m long quarter-circular section of the gate become

$$F_R = \sqrt{F_H^2 + F_V^2} = \sqrt{(313.9\ \text{kN})^2 + (134.7\ \text{kN})^2} = 341.6\ \text{kN}$$

$$\tan\theta = \frac{F_V}{F_H} = \frac{134.7\text{kN}}{313.9\ \text{kN}} = 0.429 \quad \rightarrow \quad \theta = 23.2°$$

Therefore, the magnitude of the hydrostatic force acting on the gate is 341.6 kN, and its line of action passes through the center of the quarter-circular gate making an angle 23.2° upwards from the horizontal.

The minimum spring force needed is determined by taking a moment about the point *A* where the hinge is, and setting it equal to zero,

$$\sum M_A = 0 \quad \rightarrow \quad F_R R\sin(90-\theta) - F_{\text{spring}} R = 0$$

Solving for $F_{\text{spring}}$ and substituting, the spring force is determined to be

$$F_{\text{spring}} = F_R \sin\left(90 - \theta\right) = (341.6\ \text{kN})\sin\left(90° - 23.2°\right) = \mathbf{314\ kN}$$

***Discussion*** If the previous problem is solved using a program like EES, it is simple to repeat with different values.

## Buoyancy

### 3-77C

**Solution** We are to define and discuss the buoyant force.

***Analysis*** The **upward force a fluid exerts on an immersed body** is called the *buoyant force*. The buoyant force is **caused by the increase of pressure in a fluid with depth**. The *magnitude* of the buoyant force acting on a submerged body whose volume is $V$ is expressed as $\boxed{F_B = \rho_f g V}$. The *direction* of the buoyant force is **upwards**, and its *line of action* **passes through the centroid of the displaced volume**.

***Discussion*** If the buoyant force is greater than the body's weight, it floats.

### 3-78C

**Solution** We are to compare the buoyant force on two spheres.

***Analysis*** The magnitude of the buoyant force acting on a submerged body whose volume is $V$ is expressed as $F_B = \rho_f g V$, which is independent of depth. Therefore, **the buoyant forces acting on two identical spherical balls submerged in water at different depths is the same**.

***Discussion*** Buoyant force depends only on the volume of the object, not its density.

### 3-79C

**Solution** We are to compare the buoyant force on two spheres.

***Analysis*** The magnitude of the buoyant force acting on a submerged body whose volume is $V$ is expressed as $F_B = \rho_f g V$, which is independent of the density of the body ($\rho_f$ is the fluid density). Therefore, **the buoyant forces acting on the 5-cm diameter aluminum and iron balls submerged in water is the same**.

***Discussion*** Buoyant force depends only on the volume of the object, not its density.

### 3-80C

**Solution** We are to compare the buoyant forces on a cube and a sphere.

***Analysis*** The magnitude of the buoyant force acting on a submerged body whose volume is $V$ is expressed as $F_B = \rho_f g V$, which is independent of the shape of the body. Therefore, **the buoyant forces acting on the cube and sphere made of copper submerged in water are the same since they have the same volume**.

***Discussion*** The two objects have the same volume because they have the same mass *and* density.

**3-81C**

**Solution** We are to discuss the stability of a submerged and a floating body.

***Analysis*** A *submerged body* whose **center of gravity *G* is above the center of buoyancy *B*, which is the centroid of the displaced volume, is *unstable***. But a *floating body* **may still be stable when *G* is above *B* since the centroid of the displaced volume shifts to the side to a point *B'* during a rotational disturbance while the center of gravity *G* of the body remains unchanged**. If the point *B'* is sufficiently far, these two forces create a restoring moment, and return the body to the original position.

***Discussion*** Stability analysis like this is critical in the design of ship hulls, so that they are least likely to capsize.

---

**3-82**

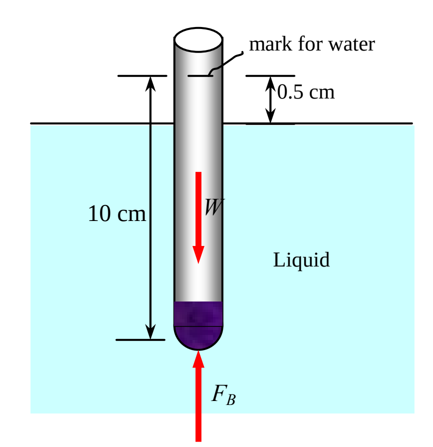
**Solution** The density of a liquid is to be determined by a hydrometer by establishing division marks in water and in the liquid, and measuring the distance between these marks.

***Properties*** We take the density of pure water to be 1000 kg/m$^3$.

***Analysis*** A hydrometer floating in water is in static equilibrium, and the buoyant force $F_B$ exerted by the liquid must always be equal to the weight $W$ of the hydrometer, $F_B = W$.

$$F_B = \rho g V_{\text{sub}} = \rho g h A_c$$

where $h$ is the height of the submerged portion of the hydrometer and $A_c$ is the cross-sectional area which is constant.




*In pure water*: $W = \rho_w g h_w A_c$

*In the liquid*: $W = \rho_{\text{liquid}} g h_{\text{liquid}} A_c$

Setting the relations above equal to each other (since both equal the weight of the hydrometer) gives

$$\rho_w g h_w A_c = \rho_{\text{liquid}} g h_{\text{liquid}} A_c$$

Solving for the liquid density and substituting,

$$\rho_{\text{liquid}} = \frac{h_{\text{water}}}{h_{\text{liquid}}}\rho_{\text{water}} = \frac{10 \text{ cm}}{(10-0.5) \text{ cm}}(1000 \text{ kg/m}^3) = 1053 \text{ kg/m}^3 \cong \mathbf{1050 \text{ kg/m}^3}$$

***Discussion*** Note that for a given cylindrical hydrometer, the product of the fluid density and the height of the submerged portion of the hydrometer is constant in any fluid.

---

**3-83E**

**Solution** A concrete block is lowered into the sea. The tension in the rope is to be determined before and after the block is immersed in water.

***Assumptions*** **1** The buoyancy force in air is negligible. **2** The weight of the rope is negligible.

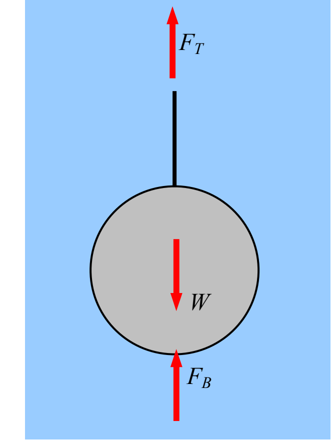
***Properties*** The density of steel block is given to be 494 lbm/ft$^3$.

***Analysis*** (*a*) The forces acting on the concrete block in air are its downward weight and the upward pull action (tension) by the rope. These two forces must balance each other, and thus the tension in the rope must be equal to the weight of the block:

$$V = 4\pi R^3/3 = 4\pi(1.5\text{ ft})^3/3 = 14.137\text{ ft}^3$$

$$F_T = W = \rho_{\text{concrete}} gV$$

$$= \left(494\text{ lbm/ft}^3\right)\left(32.2\text{ ft/s}^2\right)\left(14.137\text{ ft}^3\right)\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = 6984\text{ lbf} \cong \mathbf{6980\ lbf}$$




(*b*) When the block is immersed in water, there is the additional force of buoyancy acting upwards. The force balance in this case gives

$$F_B = \rho_f gV = \left(62.4\text{ lbm/ft}^3\right)\left(32.2\text{ ft/s}^2\right)\left(14.137\text{ ft}^3\right)\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = 882\text{ lbf}$$

$$F_{\text{T,water}} = W - F_B = 6984 - 882 = 6102\text{ lbf} \cong \mathbf{6100\ lbf}$$

***Discussion*** Note that the weight of the concrete block and thus the tension of the rope decreases by (6984 – 6102)/6984 = 12.6% in water.

---

**3-84**

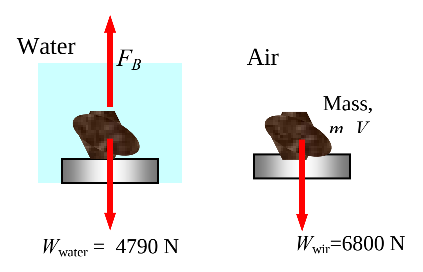
**Solution** An irregularly shaped body is weighed in air and then in water with a spring scale. The volume and the average density of the body are to be determined.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Assumptions*** **1** The buoyancy force in air is negligible. **2** The body is completely submerged in water.

***Analysis*** The mass of the body is

$$m = \frac{W_{\text{air}}}{g} = \frac{7200\text{ N}}{9.81\text{ m/s}^2}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right) = 733.9\text{ kg}$$




The difference between the weights in air and in water is due to the buoyancy force in water,

$$F_B = W_{\text{air}} - W_{\text{water}} = 7200 - 4790 = 2410\text{ N}$$

Noting that $F_B = \rho_{\text{water}} gV$ , the volume of the body is determined to be

$$V = \frac{F_B}{\rho_{\text{water}} g} = \frac{2410\text{ N}}{\left(1000\text{ kg/m}^3\right)\left(9.81\text{ m/s}^2\right)} = 0.2457\text{ m}^3 \cong \mathbf{0.246\ m^3}$$

Then the density of the body becomes

$$\rho = \frac{m}{V} = \frac{733.9\text{ kg}}{0.2457\text{ m}^3} = 2987\text{ kg/m}^3 \cong \mathbf{2990\ kg/m^3}$$

***Discussion*** The volume of the body can also be measured by observing the change in the volume of the container when the body is dropped in it (assuming the body is not porous).

---

## 3-85

**Solution** The height of the portion of a cubic ice block that extends above the water surface is measured. The height of the ice block below the surface is to be determined.

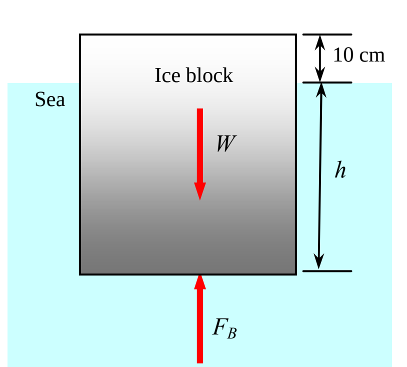
***Assumptions*** **1** The buoyancy force in air is negligible. **2** The top surface of the ice block is parallel to the surface of the sea.

***Properties*** The specific gravities of ice and seawater are given to be 0.92 and 1.025, respectively, and thus the corresponding densities are 920 kg/m$^3$ and 1025 kg/m$^3$.

***Analysis*** The weight of a body floating in a fluid is equal to the buoyant force acting on it (a consequence of vertical force balance from static equilibrium). Therefore, in this case the average density of the body must be equal to the density of the fluid since

$$W = F_B \quad \rightarrow \quad \rho_{\text{body}} g V_{\text{total}} = \rho_{\text{fluid}} g V_{\text{submerged}}$$

$$\frac{V_{\text{submerged}}}{V_{\text{total}}} = \frac{\rho_{\text{body}}}{\rho_{\text{fluid}}}$$



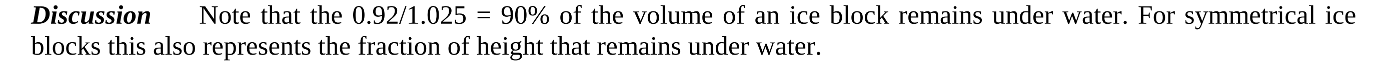

The cross-sectional of a cube is constant, and thus the "volume ratio" can be replaced by "height ratio". Then,

$$\frac{h_{\text{submerged}}}{h_{\text{total}}} = \frac{\rho_{\text{body}}}{\rho_{\text{fluid}}} \quad \rightarrow \quad \frac{h}{h+0.10} = \frac{\rho_{\text{ice}}}{\rho_{\text{water}}} \quad \rightarrow \quad \frac{h}{h+0.10} = \frac{0.92}{1.025}$$

where $h$ is the height of the ice block below the surface. Solving for $h$ gives

$$h = 0.876 \text{ m} = \mathbf{87.6 \text{ cm}}$$

***Discussion*** Note that the 0.92/1.025 = 90% of the volume of an ice block remains under water. For symmetrical ice blocks this also represents the fraction of height that remains under water.



## 3-86

**Solution** A man dives into a lake and tries to lift a large rock. The force that the man needs to apply to lift it from the bottom of the lake is to be determined.

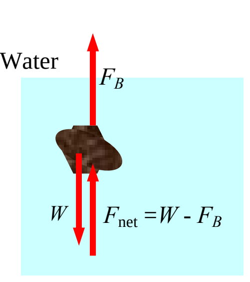
***Assumptions*** **1** The rock is c completely submerged in water. **2** The buoyancy force in air is negligible.

***Properties*** The density of granite rock is given to be 2700 kg/m$^3$. We take the density of water to be 1000 kg/m$^3$.

***Analysis*** The weight and volume of the rock are

$$W = mg = (170 \text{ kg})(9.81 \text{m/s}^2)\left(\frac{1 \text{ N}}{1 \text{kg} \cdot \text{m/s}^2}\right) = 1668 \text{ N}$$

$$V = \frac{m}{\rho} = \frac{170 \text{ kg}}{2700 \text{ kg/m}^3} = 0.06296 \text{ m}^3$$

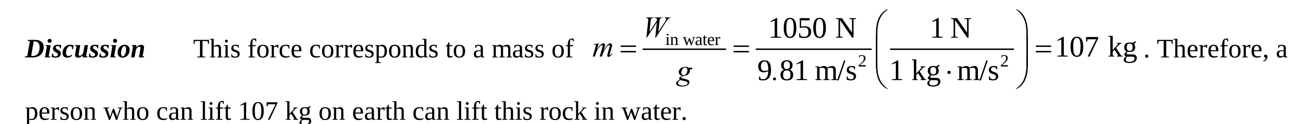



The buoyancy force acting on the rock is

$$F_B = \rho_{\text{water}} g V = \left(1000 \text{ kg/m}^3\right)\left(9.81 \text{ m/s}^2\right)\left(0.06296 \text{ m}^3\right)\left(\frac{1 \text{ N}}{1 \text{ kg} \cdot \text{m/s}^2}\right) = 618 \text{ N}$$

The weight of a body submerged in water is equal to the weigh of the body in air minus the buoyancy force,

$$W_{\text{in water}} = W_{\text{in air}} - F_B = 1668 - 618 = \mathbf{1050 \text{ N}}$$



***Discussion*** This force corresponds to a mass of $m = \frac{W_{\text{in water}}}{g} = \frac{1050 \text{ N}}{9.81 \text{ m/s}^2}\left(\frac{1 \text{ N}}{1 \text{ kg} \cdot \text{m/s}^2}\right) = 107 \text{ kg}$. Therefore, a person who can lift 107 kg on earth can lift this rock in water.

**3-87**

**Solution** An irregularly shaped crown is weighed in air and then in water with a spring scale. It is to be determined if the crown is made of pure gold.

***Assumptions*** **1** The buoyancy force in air is negligible. **2** The crown is completely submerged in water.

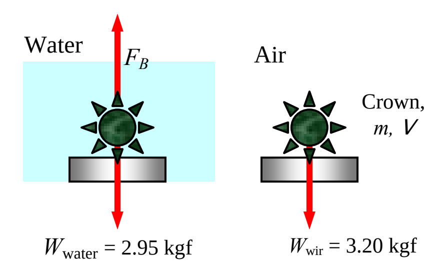
***Properties*** We take the density of water to be 1000 kg/m$^3$. The density of gold is given to be 19300 kg/m$^3$.

***Analysis*** The mass of the crown is

$$m = \frac{W_{\text{air}}}{g} = \frac{31.4\text{ N}}{9.81\text{ m/s}^2}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right) = 3.20\text{ kg}$$

The difference between the weights in air and in water is due to the buoyancy force in water, and thus

$$F_B = W_{\text{air}} - W_{\text{water}} = 31.4 - 28.9 = 2.50\text{ N}$$




Noting that $F_B = \rho_{\text{water}} g V$, the volume of the crown is determined to be

$$V = \frac{F_B}{\rho_{\text{water}} g} = \frac{2.50\text{ N}}{(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)} = 2.548\times10^{-4}\text{ m}^3$$

Then the density of the crown becomes

$$\rho = \frac{m}{V} = \frac{3.20\text{ kg}}{2.548\times10^{-4}\text{ m}^3} = 12{,}560\text{ kg/m}^3$$

which is considerably less than the density of gold. Therefore, **the crown is NOT made of pure gold**.

***Discussion*** This problem can also be solved without doing any under-water weighing as follows: We would weigh a bucket half-filled with water, and drop the crown into it. After marking the new water level, we would take the crown out, and add water to the bucket until the water level rises to the mark. We would weigh the bucket again. Dividing the weight difference by the density of water and $g$ will give the volume of the crown. Knowing both the weight and the volume of the crown, the density can easily be determined.

**3-88**

**Solution** The average density of a person is determined by weighing the person in air and then in water. A relation is to be obtained for the volume fraction of body fat in terms of densities.

***Assumptions*** **1** The buoyancy force in air is negligible. **2** The body is considered to consist of fat and muscle only. **3** The body is completely submerged in water, and the air volume in the lungs is negligible.

***Analysis*** The difference between the weights of the person in air and in water is due to the buoyancy force in water. Therefore,

$$F_B = W_{\text{air}} - W_{\text{water}} \quad \rightarrow \quad \rho_{\text{water}} g V = W_{\text{air}} - W_{\text{water}}$$

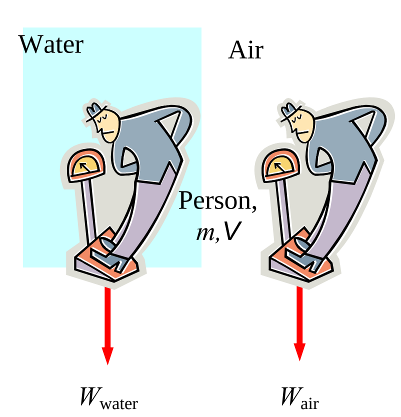
Knowing the weights and the density of water, the relation above gives the volume of the person. Then the average density of the person can be determined from

$$\rho_{\text{ave}} = \frac{m}{V} = \frac{W_{\text{air}} / g}{V}$$

Under assumption #2, the total mass of a person is equal to the sum of the masses of the fat and muscle tissues, and the total volume of a person is equal to the sum of the volumes of the fat and muscle tissues. The volume fraction of body fat is the ratio of the fat volume to the total volume of the person. Therefore,

$$V = V_{\text{fat}} + V_{\text{muscle}} \quad \text{where} \quad V_{\text{fat}} = x_{\text{fat}} V \quad \text{and} \quad V_{\text{muscle}} = x_{\text{muscle}} V = (1 - x_{\text{fat}}) V$$

$$m = m_{\text{fat}} + m_{\text{muscle}}$$




Noting that mass is density times volume, the last relation can be written as

$$\rho_{\text{ave}} V = \rho_{\text{fat}} V_{\text{fat}} + \rho_{\text{muscle}} V_{\text{muscle}}$$

$$\rho_{\text{ave}} V = \rho_{\text{fat}} x_{\text{fat}} V + \rho_{\text{muscle}} (1 - x_{\text{fat}}) V$$

Canceling the $V$ and solving for $x_{\text{fat}}$ gives the desired relation,

$$\boxed{x_{\text{fat}} = \frac{\rho_{\text{muscle}} - \rho_{\text{avg}}}{\rho_{\text{muscle}} - \rho_{\text{fat}}}}$$

***Discussion*** Weighing a person in water in order to determine its volume is not practical. A more practical way is to use a large container, and measuring the change in volume when the person is completely submerged in it.

---

**3-89**

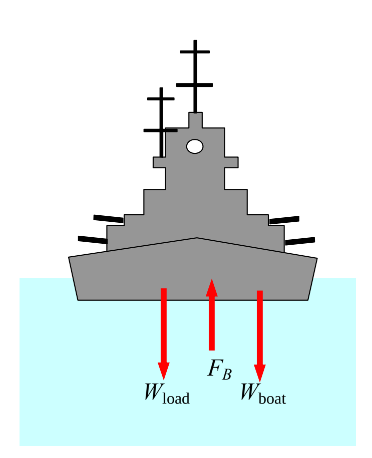
**Solution** The volume of the hull of a boat is given. The amounts of load the boat can carry in a lake and in the sea are to be determined.

***Assumptions*** **1** The dynamic effects of the waves are disregarded. **2** The buoyancy force in air is negligible.

***Properties*** The density of sea water is given to be 1.03×1000 = 1030 kg/m$^3$. We take the density of water to be 1000 kg/m$^3$.

***Analysis*** The weight of the unloaded boat is

$$W_{\text{boat}} = mg = (8560\text{ kg})(9.81\text{ m/s}^2)\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 84.0\text{ kN}$$




The buoyancy force becomes a maximum when the entire hull of the boat is submerged in water, and is determined to be

$$F_{B,\text{lake}} = \rho_{\text{lake}} g V = (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(150\text{ m}^3)\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 1472\text{ kN}$$

$$F_{B,\text{sea}} = \rho_{\text{sea}} g V = (1030\text{ kg/m}^3)(9.81\text{ m/s}^2)(150\text{ m}^3)\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 1516\text{ kN T}$$

he total weight of a floating boat (load + boat itself) is equal to the buoyancy force. Therefore, the weight of the maximum load is

$$W_{\text{load, lake}} = F_B\text{, lake} - W_{\text{boat}} = 1472 - 84 = 1388\text{ kN}$$

$$W_{\text{load, sea}} = F_{B,\text{sea}} - W_{\text{boat}} = 1516 - 84 = 1432\text{ kN}$$



The corresponding masses of load are

$$m_{\text{load,lake}} = \frac{W_{\text{load,lake}}}{g} = \frac{1388\text{ kN}}{9.81\text{ m/s}^2}\left(\frac{1000\text{ kg}\cdot\text{m/s}^2}{1\text{ kN}}\right) = 141{,}500\text{ kg} \cong \mathbf{142{,}000\text{ kg}}$$

$$m_{\text{load,sea}} = \frac{W_{\text{load,lsea}}}{g} = \frac{1432\text{ kN}}{9.81\text{ m/s}^2}\left(\frac{1000\text{ kg}\cdot\text{m/s}^2}{1\text{ kN}}\right) = 145{,}970\text{ kg} \cong \mathbf{146{,}000\text{ kg}}$$

***Discussion*** Note that this boat can carry nearly 4500 kg more load in the sea than it can in fresh water. Fully-loaded boats in sea water should expect to sink into water deeper when they enter fresh water, such as a river where the port may be.

## Fluids in Rigid-Body Motion

**3-90C**

**Solution** We are to discuss when a fluid can be treated as a rigid body.

***Analysis*** A moving body of fluid can be treated as a rigid body **when there are no shear stresses (i.e., no motion between fluid layers relative to each other) in the fluid body**.

***Discussion*** When there is no relative motion between fluid particles, there are no viscous stresses, and pressure (normal stress) is the only stress.

**3-91C**

**Solution** We are to compare the pressure at the bottom of a glass of water moving at various velocities.

***Analysis*** The water pressure at the bottom surface is **the same for all cases** since the acceleration for all four cases is zero.

***Discussion*** When any body, fluid or solid, moves at constant velocity, there is no acceleration, regardless of the direction of the movement.

**3-92C**

**Solution** We are to compare the pressure in a glass of water for stationary and accelerating conditions.

***Analysis*** The pressure at the bottom surface is constant when the glass is stationary. For a glass moving on a horizontal plane with constant acceleration, water will collect at the back but the water depth will remain constant at the center. Therefore, the pressure at the midpoint will be the same for both glasses. But **the bottom pressure will be low at the front relative to the stationary glass, and high at the back** (again relative to the stationary glass). Note that the pressure in all cases is the hydrostatic pressure, which is directly proportional to the fluid height.

***Discussion*** We ignore any sloshing of the water.

**3-93C**

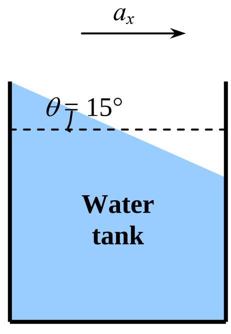
**Solution** We are to analyze the pressure in a glass of water that is rotating.

***Analysis*** When a vertical cylindrical container partially filled with water is rotated about its axis and rigid body motion is established, the fluid level will drop at the center and rise towards the edges. Noting that hydrostatic pressure is proportional to fluid depth, **the pressure at the mid point will drop and the pressure at the edges of the bottom surface will rise due to the rotation**.

***Discussion*** The highest pressure occurs at the bottom corners of the container.

## 3-94

**Solution** A water tank is being towed by a truck on a level road, and the angle the free surface makes with the horizontal is measured. The acceleration of the truck is to be determined.




***Assumptions*** **1** The road is horizontal so that acceleration has no vertical component ($a_z$ = 0). **2** Effects of splashing, breaking, driving over bumps, and climbing hills are assumed to be secondary, and are not considered. **3** The acceleration remains constant.

***Analysis*** We take the $x$-axis to be the direction of motion, the $z$-axis to be the upward vertical direction. The tangent of the angle the free surface makes with the horizontal is

$$\tan\theta = \frac{a_x}{g + a_z}$$

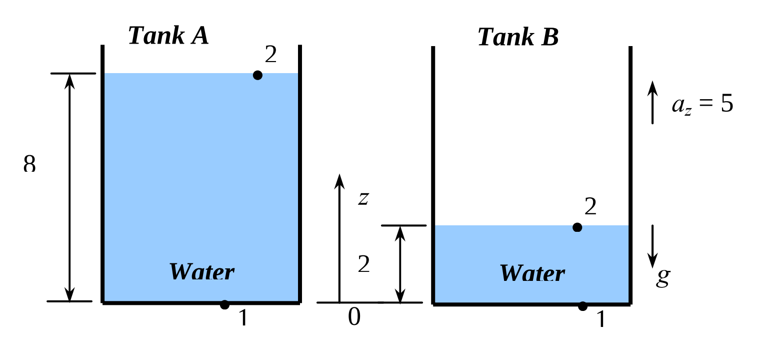
Solving for $a_x$ and substituting,

$$a_x = (g + a_z)\tan\theta = (9.81\ \text{m/s}^2 + 0)\tan 15° = \mathbf{2.63\ \ m/s^2}$$

***Discussion*** Note that the analysis is valid for any fluid with constant density since we used no information that pertains to fluid properties in the solution.

## 3-95

**Solution** Two water tanks filled with water, one stationary and the other moving upwards at constant acceleration. The tank with the higher pressure at the bottom is to be determined.




***Assumptions*** **1** The acceleration remains constant. **2** Water is an incompressible substance.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** The pressure difference between two points 1 and 2 in an incompressible fluid is given by

$$P_2 - P_1 = -\rho a_x(x_2 - x_1) - \rho(g + a_z)(z_2 - z_1) \qquad \text{or} \qquad P_1 - P_2 = \rho(g + a_z)(z_2 - z_1)$$

since $a_x$ = 0. Taking point 2 at the free surface and point 1 at the tank bottom, we have $P_2 = P_{atm}$ and $z_2 - z_1 = h$ and thus

$$P_{1,\text{ gage}} = P_{\text{bottom}} = \rho(g + a_z)h$$

**Tank A**: We have $a_z$ = 0, and thus the pressure at the bottom is

$$P_{A,\text{ bottom}} = \rho g h_A = (1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(8\ \text{m})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) = 78.5\ \text{kN/m}^2$$

**Tank B**: We have $a_z$ = +5 m/s$^2$, and thus the pressure at the bottom is

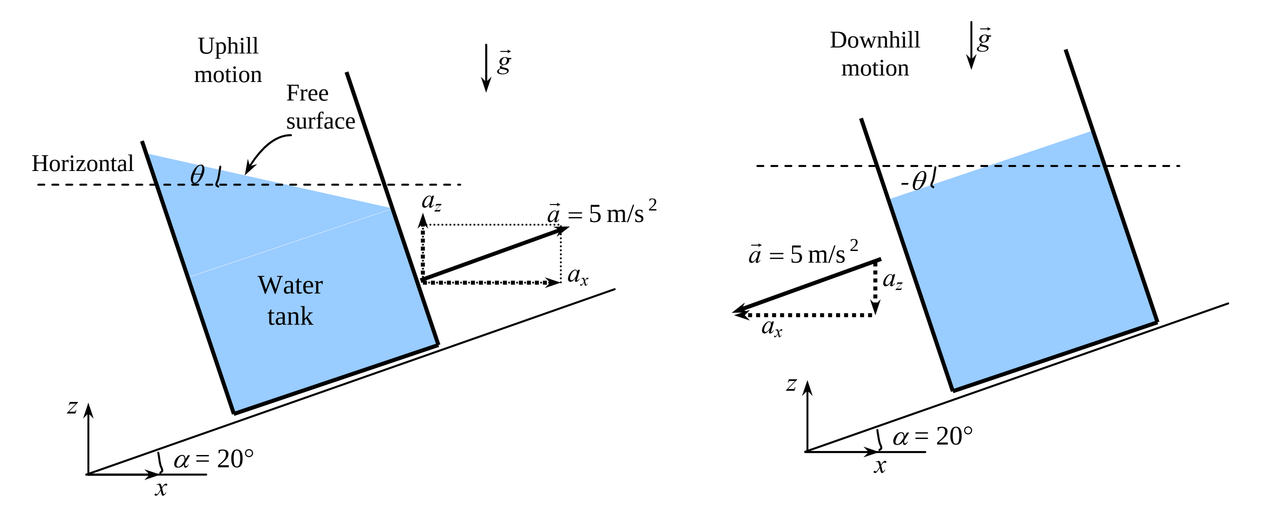
$$P_{B,\text{ bottom}} = \rho(g + a_z)h_B = (1000\ \text{kg/m}^3)(9.81 + 5\ \text{m/s}^2)(2\ \text{m})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) = 29.6\ \text{kN/m}^2$$

Therefore, **tank A has a higher pressure at the bottom**.

***Discussion*** We can also solve this problem quickly by examining the relation $P_{\text{bottom}} = \rho(g + a_z)h$. Acceleration for tank B is about 1.5 times that of Tank A (14.81 vs 9.81 m/s$^2$), but the fluid depth for tank A is 4 times that of tank B (8 m vs 2 m). Therefore, the tank with the larger acceleration-fluid height product (tank A in this case) will have a higher pressure at the bottom.

**3-96**

**Solution** A water tank is being towed on an uphill road at constant acceleration. The angle the free surface of water makes with the horizontal is to be determined, and the solution is to be repeated for the downhill motion case.




***Assumptions*** **1** Effects of splashing, breaking, driving over bumps, and climbing hills are assumed to be secondary, and are not considered. **2** The acceleration remains constant.

***Analysis*** We take the *x*- and *z*-axes as shown in the figure. From geometrical considerations, the horizontal and vertical components of acceleration are

$$a_x = a\cos\alpha$$
$$a_z = a\sin\alpha$$

The tangent of the angle the free surface makes with the horizontal is

$$\tan\theta = \frac{a_x}{g+a_z} = \frac{a\cos\alpha}{g+a\sin\alpha} = \frac{(5\ \mathrm{m/s^2})\cos 20°}{9.81\ \mathrm{m/s^2} + (5\ \mathrm{m/s^2})\sin 20°} = 0.4078 \quad\rightarrow\quad \theta = \mathbf{22.2°}$$

When the direction of motion is reversed, both $a_x$ and $a_z$ are in negative *x*- and *z*-direction, respectively, and thus become negative quantities,

$$a_x = -a\cos\alpha$$
$$a_z = -a\sin\alpha$$

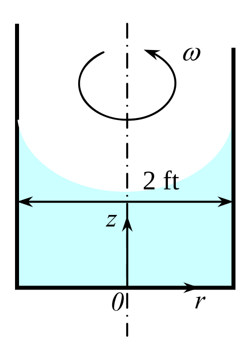
Then the tangent of the angle the free surface makes with the horizontal becomes

$$\tan\theta = \frac{a_x}{g+a_z} = \frac{a\cos\alpha}{g+a\sin\alpha} = \frac{-(5\ \mathrm{m/s^2})\cos 20°}{9.81\ \mathrm{m/s^2} - (5\ \mathrm{m/s^2})\sin 20°} = -0.5801 \quad\rightarrow\quad \theta = \mathbf{-\ 30.1°}$$

***Discussion*** Note that the analysis is valid for any fluid with constant density, not just water, since we used no information that pertains to water in the solution.

**3-97E**

**Solution** A vertical cylindrical tank open to the atmosphere is rotated about the centerline. The angular velocity at which the bottom of the tank will first be exposed, and the maximum water height at this moment are to be determined.




***Assumptions*** **1** The increase in the rotational speed is very slow so that the liquid in the container always acts as a rigid body. **2** Water is an incompressible fluid.

***Analysis*** Taking the center of the bottom surface of the rotating vertical cylinder as the origin ($r$ = 0, $z$ = 0), the equation for the free surface of the liquid is given as

$$z_s(r) = h_0 - \frac{\omega^2}{4g}(R^2 - 2r^2)$$

where $h_0$ = 1 ft is the original height of the liquid before rotation. Just before dry spot appear at the center of bottom surface, the height of the liquid at the center equals zero, and thus $z_s(0)$ = 0. Solving the equation above for ω and substituting,

$$\omega = \sqrt{\frac{4gh_0}{R^2}} = \sqrt{\frac{4\left(32.2 \text{ ft/s}^2\right)\left(1 \text{ ft}\right)}{\left(1 \text{ ft}\right)^2}} = 11.35 \text{ rad/s} \cong \mathbf{11.4 \text{ rad/s}}$$

Noting that one complete revolution corresponds to 2π radians, the rotational speed of the container can also be expressed in terms of revolutions per minute (rpm) as

$$\dot{n} = \frac{\omega}{2\pi} = \frac{11.35 \text{ rad/s}}{2\pi \text{ rad/rev}}\left(\frac{60 \text{ s}}{1 \text{ min}}\right) = \mathbf{108 \text{ rpm}}$$

Therefore, the rotational speed of this container should be limited to 108 rpm to avoid any dry spots at the bottom surface of the tank.

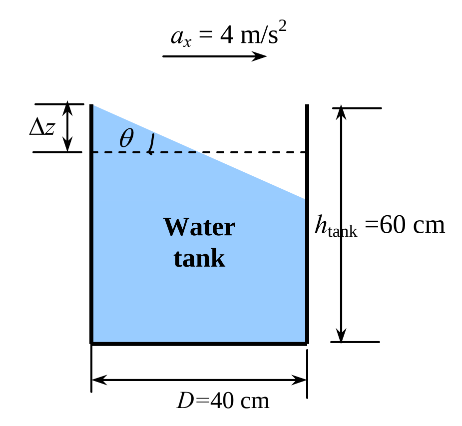
The maximum vertical height of the liquid occurs a the edges of the tank ($r = R$ = 1 ft), and it is

$$z_s(R) = h_0 + \frac{\omega^2 R^2}{4g} = (1\text{ ft}) + \frac{(11.35 \text{ rad/s})^2 (1\text{ ft})^2}{4(32.2 \text{ ft/s}^2)} = \mathbf{2.00 \text{ ft}}$$

***Discussion*** Note that the analysis is valid for any liquid since the result is independent of density or any other fluid property.

**3-98**

**Solution** A cylindrical tank is being transported on a level road at constant acceleration. The allowable water height to avoid spill of water during acceleration is to be determined.




***Assumptions*** **1** The road is horizontal during acceleration so that acceleration has no vertical component ($a_z$ = 0). **2** Effects of splashing, breaking, driving over bumps, and climbing hills are assumed to be secondary, and are not considered. **3** The acceleration remains constant.

***Analysis*** We take the *x*-axis to be the direction of motion, the *z*-axis to be the upward vertical direction, and the origin to be the midpoint of the tank bottom. The tangent of the angle the free surface makes with the horizontal is

$$\tan\theta = \frac{a_x}{g + a_z} = \frac{4}{9.81 + 0} = 0.4077 \quad \text{(and thus } \theta = 22.2°\text{)}$$

The maximum vertical rise of the free surface occurs at the back of the tank, and the vertical midplane experiences no rise or drop during acceleration. Then the maximum vertical rise at the back of the tank relative to the midplane is

$$\Delta z_{\max} = (D/2)\tan\theta = [(0.40\text{ m})/2] \times 0.4077 = 0.082\text{ m} = 8.2\text{ cm}$$

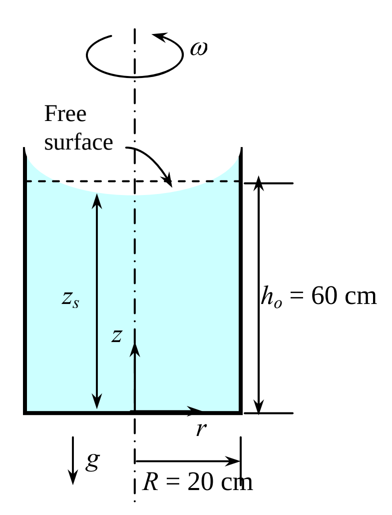
Therefore, the maximum initial water height in the tank to avoid spilling is

$$h_{\max} = h_{\text{tank}} - \Delta z_{\max} = 60 - 8.2 = \mathbf{51.8\ cm}$$

***Discussion*** Note that the analysis is valid for any fluid with constant density, not just water, since we used no information that pertains to water in the solution.

**3-99**

**Solution** A vertical cylindrical container partially filled with a liquid is rotated at constant speed. The drop in the liquid level at the center of the cylinder is to be determined.




***Assumptions*** **1** The increase in the rotational speed is very slow so that the liquid in the container always acts as a rigid body. **2** The bottom surface of the container remains covered with liquid during rotation (no dry spots).

***Analysis*** Taking the center of the bottom surface of the rotating vertical cylinder as the origin ($r$ = 0, $z$ = 0), the equation for the free surface of the liquid is given as

$$z_s(r) = h_0 - \frac{\omega^2}{4g}(R^2 - 2r^2)$$

where $h_0$ = 0.6 m is the original height of the liquid before rotation, and

$$\omega = 2\pi\dot{n} = 2\pi(120 \text{ rev/min})\left(\frac{1 \text{ min}}{60 \text{ s}}\right) = 12.57 \text{ rad/s}$$

Then the vertical height of the liquid at the center of the container where $r$ = 0 becomes

$$z_s(0) = h_0 - \frac{\omega^2 R^2}{4g} = (0.60 \text{ m}) - \frac{(12.57 \text{ rad/s})^2 (0.20 \text{ m})^2}{4(9.81 \text{ m/s}^2)} = 0.44 \text{ m}$$

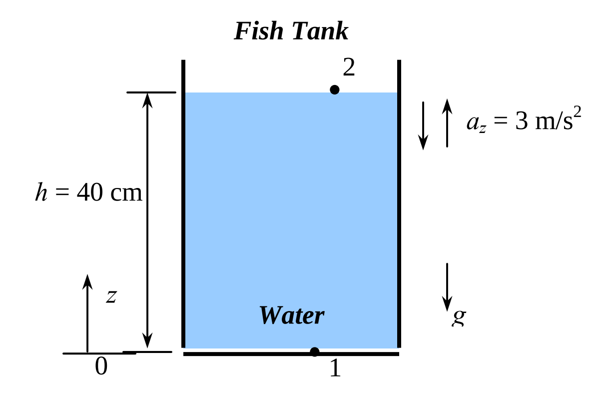
Therefore, the drop in the liquid level at the center of the cylinder is

$$\Delta h_{\text{drop, center}} = h_0 - z_s(0) = 0.60 - 0.44 = \mathbf{0.16 \ m}$$

***Discussion*** Note that the analysis is valid for any liquid since the result is independent of density or any other fluid property. Also, our assumption of no dry spots is validated since $z_0(0)$ is positive.

**3-100**

**Solution** The motion of a fish tank in the cabin of an elevator is considered. The pressure at the bottom of the tank when the elevator is stationary, moving up with a specified acceleration, and moving down with a specified acceleration is to be determined.




***Assumptions*** **1** The acceleration remains constant. **2** Water is an incompressible substance.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** The pressure difference between two points 1 and 2 in an incompressible fluid is given by

$$P_2 - P_1 = -\rho a_x(x_2 - x_1) - \rho(g + a_z)(z_2 - z_1) \quad \text{or} \quad P_1 - P_2 = \rho(g + a_z)(z_2 - z_1)$$

since $a_x = 0$. Taking point 2 at the free surface and point 1 at the tank bottom, we have $P_2 = P_{atm}$ and $z_2 - z_1 = h$ and thus

$$P_{1,\text{gage}} = P_{\text{bottom}} = \rho(g + a_z)h$$

(***a***) **Tank stationary**: We have $a_z = 0$, and thus the gage pressure at the tank bottom is

$$P_{\text{bottom}} = \rho g h = (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.4\text{ m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 3.92\text{ kN/m}^2 = \mathbf{3.92\ kPa}$$

(***b***) **Tank moving up**: We have $a_z = +3$ m/s$^2$, and thus the gage pressure at the tank bottom is

$$P_{\text{bottom}} = \rho(g + a_z)h_B = (1000\text{ kg/m}^3)(9.81 + 3\text{ m/s}^2)(0.4\text{ m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 5.12\text{ kN/m}^2 = \mathbf{5.12\ kPa}$$

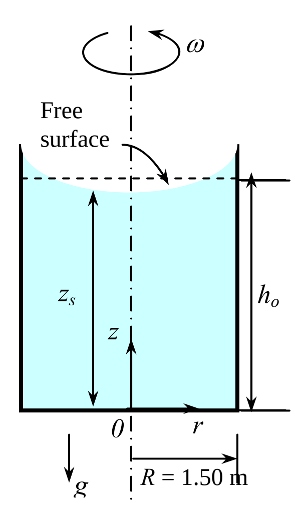
(***c***) **Tank moving down**: We have $a_z = -3$ m/s$^2$, and thus the gage pressure at the tank bottom is

$$P_{\text{bottom}} = \rho(g + a_z)h_B = (1000\text{ kg/m}^3)(9.81 - 3\text{ m/s}^2)(0.4\text{ m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 2.72\text{ kN/m}^2 = \mathbf{2.72\ kPa}$$

***Discussion*** Note that the pressure at the tank bottom while moving up in an elevator is almost twice that while moving down, and thus the tank is under much greater stress during upward acceleration.

**3-101**

**Solution** A vertical cylindrical milk tank is rotated at constant speed, and the pressure at the center of the bottom surface is measured. The pressure at the edge of the bottom surface is to be determined.




***Assumptions*** **1** The increase in the rotational speed is very slow so that the liquid in the container always acts as a rigid body. **2** Milk is an incompressible substance.

***Properties*** The density of the milk is given to be 1030 kg/m$^3$.

***Analysis*** Taking the center of the bottom surface of the rotating vertical cylinder as the origin ($r$ = 0, $z$ = 0), the equation for the free surface of the liquid is given as

$$z_s(r) = h_0 - \frac{\omega^2}{4g}(R^2 - 2r^2)$$

where $R$ = 1.5 m is the radius, and

$$\omega = 2\pi\dot{n} = 2\pi(12\text{ rev/min})\left(\frac{1\text{ min}}{60\text{ s}}\right) = 1.2566\text{ rad/s}$$

The fluid rise at the edge relative to the center of the tank is

$$\Delta h = z_s(R) - z_s(0) = \left(h_0 + \frac{\omega^2 R^2}{4g}\right) - \left(h_0 - \frac{\omega^2 R^2}{4g}\right) = \frac{\omega^2 R^2}{2g} = \frac{(1.2566\text{ rad/s})^2(1.50\text{ m})^2}{2(9.81\text{ m/s}^2)} = 1.1811\text{ m}$$

The pressure difference corresponding to this fluid height difference is

$$\Delta P_{\text{bottom}} = \rho g \Delta h = (1030\text{ kg/m}^3)(9.81\text{ m/s}^2)(1.1811\text{ m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 1.83\text{ kN/m}^2 = 1.83\text{ kPa}$$

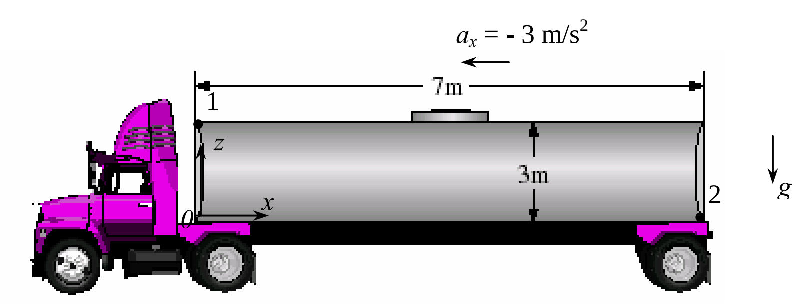
Then the pressure at the edge of the bottom surface becomes

$$P_{\text{bottom, edge}} = P_{\text{bottom, center}} + \Delta P_{\text{bottom}} = 130 + 1.83 = 131.83\text{ kPa} \cong \mathbf{132\text{ kPa}}$$

***Discussion*** Note that the pressure is 1.4% higher at the edge relative to the center of the tank, and there is a fluid level difference of 1.18 m between the edge and center of the tank, and these differences should be considered when designing rotating fluid tanks.

**3-102**

**Solution** Milk is transported in a completely filled horizontal cylindrical tank accelerating at a specified rate. The maximum pressure difference in the tanker is to be determined.




***Assumptions*** **1** The acceleration remains constant. **2** Milk is an incompressible substance.

***Properties*** The density of the milk is given to be 1020 kg/m³.

***Analysis*** We take the $x$- and $z$- axes as shown. The horizontal acceleration is in the negative $x$ direction, and thus $a_x$ is negative. Also, there is no acceleration in the vertical direction, and thus $a_z = 0$. The pressure difference between two points 1 and 2 in an incompressible fluid in linear rigid body motion is given by

$$P_2 - P_1 = -\rho a_x(x_2 - x_1) - \rho(g + a_z)(z_2 - z_1) \quad \rightarrow \quad P_2 - P_1 = -\rho a_x(x_2 - x_1) - \rho g(z_2 - z_1)$$

The first term is due to acceleration in the horizontal direction and the resulting compression effect towards the back of the tanker, while the second term is simply the hydrostatic pressure that increases with depth. Therefore, we reason that the lowest pressure in the tank will occur at point 1 (upper front corner), and the higher pressure at point 2 (the lower rear corner). Therefore, the maximum pressure difference in the tank is

$$\Delta P_{\max} = P_2 - P_1 = -\rho a_x(x_2 - x_1) - \rho g(z_2 - z_1) = -[a_x(x_2 - x_1) + g(z_2 - z_1)]$$

$$= -(1020\ \text{kg/m}^3)\left[(-2.5\ \text{m/s}^2)(7\ \text{m}) + (9.81\ \text{m/s}^2)(-3\ \text{m})\right]\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)$$

$$= (17.9 + 30.0)\ \text{kN/m}^2 = \mathbf{47.9\ kPa}$$

since $x_1 = 0$, $x_2 = 7$ m, $z_1 = 3$ m, and $z_2 = 0$.

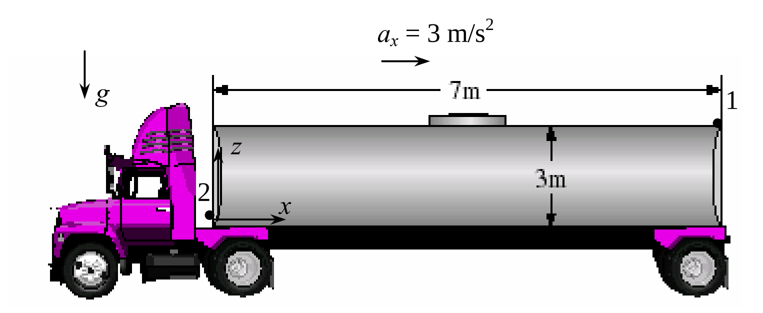
***Discussion*** Note that the variation of pressure along a horizontal line is due to acceleration in the horizontal direction while the variation of pressure in the vertical direction is due to the effects of gravity and acceleration in the vertical direction (which is zero in this case).

**3-103**

**Solution** Milk is transported in a completely filled horizontal cylindrical tank decelerating at a specified rate. The maximum pressure difference in the tanker is to be determined.

***Assumptions*** **1** The acceleration remains constant. **2** Milk is an incompressible substance.

***Properties*** The density of the milk is given to be 1020 kg/m³.




***Analysis*** We take the *x*- and *z*- axes as shown. The horizontal deceleration is in the *x* direction, and thus $a_x$ is positive. Also, there is no acceleration in the vertical direction, and thus $a_z = 0$. The pressure difference between two points 1 and 2 in an incompressible fluid in linear rigid body motion is given by

$$P_2 - P_1 = -\rho a_x (x_2 - x_1) - \rho(g + a_z)(z_2 - z_1) \quad \rightarrow \quad P_2 - P_1 = -\rho a_x (x_2 - x_1) - \rho g(z_2 - z_1)$$

The first term is due to deceleration in the horizontal direction and the resulting compression effect towards the front of the tanker, while the second term is simply the hydrostatic pressure that increases with depth. Therefore, we reason that the lowest pressure in the tank will occur at point 1 (upper front corner), and the higher pressure at point 2 (the lower rear corner). Therefore, the maximum pressure difference in the tank is

$$\Delta P_{\max} = P_2 - P_1 = -\rho a_x (x_2 - x_1) - \rho g(z_2 - z_1) = -[a_x (x_2 - x_1) + g(z_2 - z_1)]$$

$$= -(1020 \text{ kg/m}^3)\left[(2.5 \text{ m/s}^2)(-7 \text{ m}) + (9.81 \text{ m/s}^2)(-3 \text{ m})\right]\left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right)$$

$$= (17.9 + 30.0) \text{ kN/m}^2 = \mathbf{47.9 \text{ kPa}}$$

since $x_1$ = 7 m, $x_2$ = 0, $z_1$ = 3 m, and $z_2$ = 0.

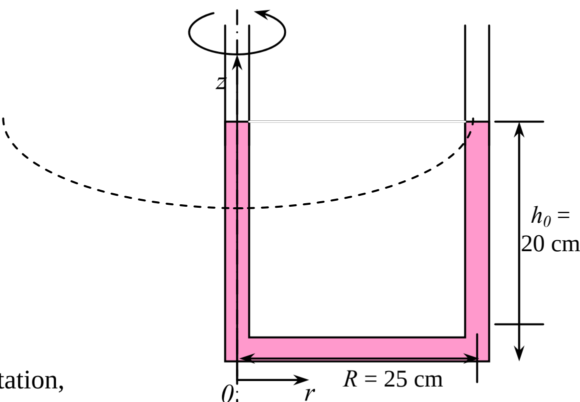
***Discussion*** Note that the variation of pressure along a horizontal line is due to acceleration in the horizontal direction while the variation of pressure in the vertical direction is due to the effects of gravity and acceleration in the vertical direction (which is zero in this case).

---

**3-104**

**Solution** A vertical U-tube partially filled with alcohol is rotated at a specified rate about one of its arms. The elevation difference between the fluid levels in the two arms is to be determined.

***Assumptions*** **1** Alcohol is an incompressible fluid.




***Analysis*** Taking the base of the left arm of the U-tube as the origin ($r$ = 0, $z$ = 0), the equation for the free surface of the liquid is given as

$$z_s(r) = h_0 - \frac{\omega^2}{4g}(R^2 - 2r^2)$$

where $h_0$ = 0.20 m is the original height of the liquid before rotation, and ω = 4.2 rad/s. The fluid rise at the right arm relative to the fluid level in the left arm (the center of rotation) is

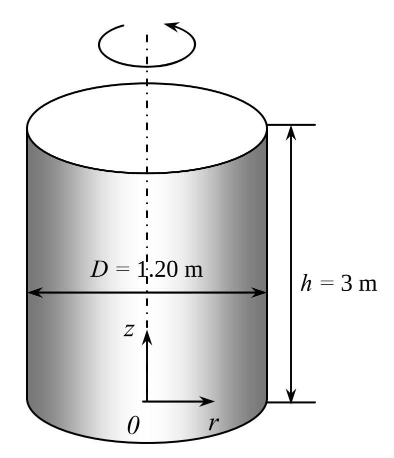
$$\Delta h = z_s(R) - z_s(0) = \left(h_0 + \frac{\omega^2 R^2}{4g}\right) - \left(h_0 - \frac{\omega^2 R^2}{4g}\right) = \frac{\omega^2 R^2}{2g} = \frac{(4.2 \text{ rad/s})^2 (0.25 \text{ m})^2}{2(9.81 \text{ m/s}^2)} = \mathbf{0.056 \text{ m}}$$

***Discussion*** The analysis is valid for any liquid since the result is independent of density or any other fluid property.

---

**3-105**

**Solution** A vertical cylindrical tank is completely filled with gasoline, and the tank is rotated about its vertical axis at a specified rate. The pressures difference between the centers of the bottom and top surfaces, and the pressures difference between the center and the edge of the bottom surface are to be determined.




***Assumptions*** **1** The increase in the rotational speed is very slow so that the liquid in the container always acts as a rigid body. **2** Gasoline is an incompressible substance.

***Properties*** The density of the gasoline is given to be 740 kg/m$^3$.

***Analysis*** The pressure difference between two points 1 and 2 in an incompressible fluid rotating in rigid body motion is given by

$$P_2 - P_1 = \frac{\rho\omega^2}{2}(r_2^2 - r_1^2) - \rho g(z_2 - z_1)$$

where $R$ = 0.60 m is the radius, and

$$\omega = 2\pi\dot{n} = 2\pi(70 \text{ rev/min})\left(\frac{1\text{ min}}{60\text{ s}}\right) = 7.330 \text{ rad/s}$$

(*a*) Taking points 1 and 2 to be the centers of the bottom and top surfaces, respectively, we have $r_1 = r_2 = 0$ and $z_2 - z_1 = h = 3\text{ m}$. Then,

$$P_{\text{center, top}} - P_{\text{center, bottom}} = 0 - \rho g(z_2 - z_1) = -\rho g h$$

$$= -(740 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(3\text{ m})\left(\frac{1\text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right) = 21.8 \text{ kN/m}^2 = \mathbf{21.8\ kPa}$$

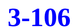
(*b*) Taking points 1 and 2 to be the center and edge of the bottom surface, respectively, we have $r_1 = 0$, $r_2 = R$, and $z_2 = z_1 = 0$. Then,

$$P_{\text{edge, bottom}} - P_{\text{center, bottom}} = \frac{\rho\omega^2}{2}(R_2^2 - 0) - 0 = \frac{\rho\omega^2 R^2}{2}$$

$$= \frac{(740 \text{ kg/m}^3)(7.33 \text{ rad/s})^2(0.60\text{ m})^2}{2}\left(\frac{1\text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right) = 7.16 \text{ kN/m}^2 = \mathbf{7.16\ kPa}$$

***Discussion*** Note that the rotation of the tank does not affect the pressure difference along the axis of the tank. But the pressure difference between the edge and the center of the bottom surface (or any other horizontal plane) is due entirely to the rotation of the tank.

---


**3-106** 

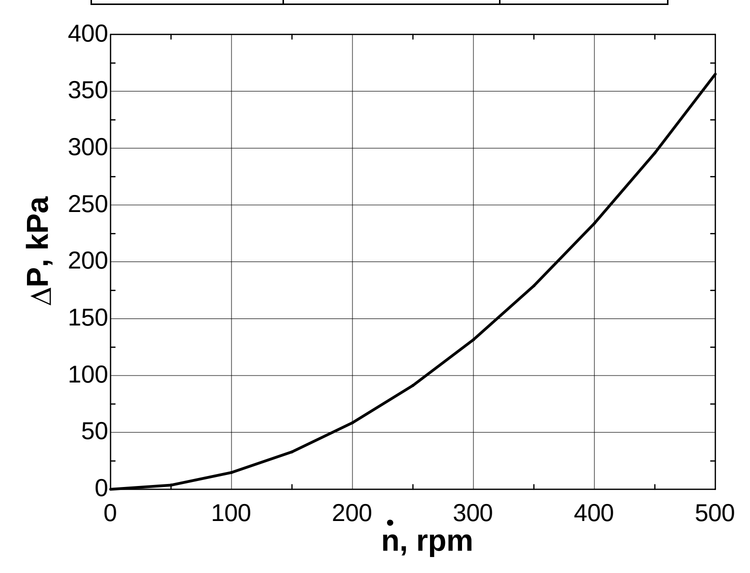
**Solution** The previous problem is reconsidered. The effect of rotational speed on the pressure difference between the center and the edge of the bottom surface of the cylinder as the rotational speed varies from 0 to 500 rpm in increments of 50 rpm is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
g=9.81 "m/s2"
rho=740 "kg/m3"
R=0.6 "m"
h=3 "m"

omega=2*pi*n_dot/60 "rad/s"
DeltaP_axis=rho*g*h/1000 "kPa"
DeltaP_bottom=rho*omega^2*R^2/2000 "kPa"
```

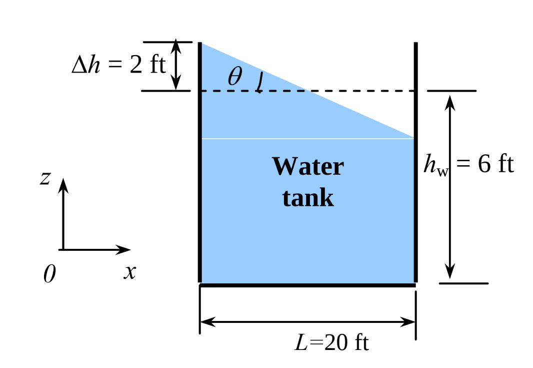
| Rotation rate $\dot{n}$ , rpm | Angular speed $\omega$, rad/s | $\Delta P_{\text{center-edge}}$ kPa |
|---|---|---|
| 0 | 0.0 | 0.0 |
| 50 | 5.2 | 3.7 |
| 100 | 10.5 | 14.6 |
| 150 | 15.7 | 32.9 |
| 200 | 20.9 | 58.4 |
| 250 | 26.2 | 91.3 |
| 300 | 31.4 | 131.5 |
| 350 | 36.7 | 178.9 |
| 400 | 41.9 | 233.7 |
| 450 | 47.1 | 295.8 |
| 500 | 52.4 | 365.2 |




***Discussion*** The pressure rise with rotation rate is not linear, but rather quadratic.

**3-107**

**Solution** A water tank partially filled with water is being towed by a truck on a level road. The maximum acceleration (or deceleration) of the truck to avoid spilling is to be determined.




***Assumptions*** **1** The road is horizontal so that acceleration has no vertical component ($a_z$ = 0). **2** Effects of splashing, breaking, driving over bumps, and climbing hills are assumed to be secondary, and are not considered. **3** The acceleration remains constant.

***Analysis*** We take the *x*-axis to be the direction of motion, the *z*-axis to be the upward vertical direction. The shape of the free surface just before spilling is shown in figure. The tangent of the angle the free surface makes with the horizontal is given by

$$\tan\theta = \frac{a_x}{g + a_z} \qquad \rightarrow \qquad a_x = g\tan\theta$$

where $a_z$ = 0 and, from geometric considerations, tan$\theta$ is $\tan\theta = \dfrac{\Delta h}{L/2}$ . Substituting, we get

$$a_x = g\tan\theta = g\frac{\Delta h}{L/2} = (32.2\text{ ft/s}^2)\frac{2\text{ ft}}{(20\text{ ft})/2} = \mathbf{6.44\ m/s^2}$$

The solution can be repeated for deceleration by replacing $a_x$ by – $a_x$. We obtain $a_x$ = **-6.44 m/s²**.

***Discussion*** Note that the analysis is valid for any fluid with constant density since we used no information that pertains to fluid properties in the solution.

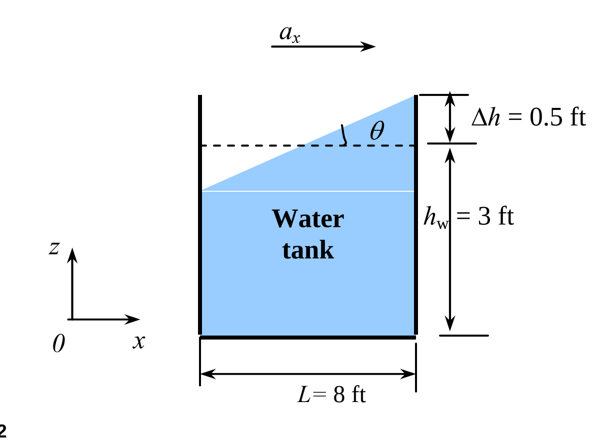
---

**3-108E**

**Solution** A water tank partially filled with water is being towed by a truck on a level road. The maximum acceleration (or deceleration) of the truck to avoid spilling is to be determined.

***Assumptions*** **1** The road is horizontal so that deceleration has no vertical component ($a_z$ = 0). **2** Effects of splashing and driving over bumps are assumed to be secondary, and are not considered. **3** The deceleration remains constant.

***Analysis*** We take the *x*-axis to be the direction of motion, the *z*-axis to be the upward vertical direction. The shape of the free surface just before spilling is shown in figure. The tangent of the angle the free surface makes with the horizontal is given by




$$\tan\theta = \frac{-a_x}{g + a_z} \qquad \rightarrow \qquad a_x = -g\tan\theta$$

where $a_z$ = 0 and, from geometric considerations, tan$\theta$ is

$$\tan\theta = \frac{\Delta h}{L/2}$$

Substituting,

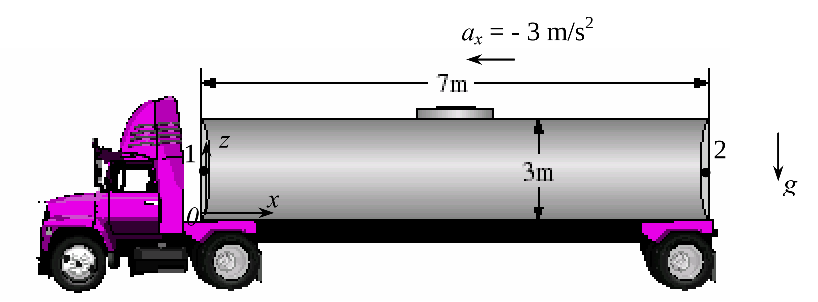
$$a_x = -g\tan\theta = -g\frac{\Delta h}{L/2} = -(32.2\text{ ft/s}^2)\frac{0.5\text{ ft}}{(8\text{ ft})/2} = \mathbf{-4.08\ ft/s^2}$$

***Discussion*** Note that the analysis is valid for any fluid with constant density since we used no information that pertains to fluid properties in the solution.

---

**3-109**

**Solution** Water is transported in a completely filled horizontal cylindrical tanker accelerating at a specified rate. The pressure difference between the front and back ends of the tank along a horizontal line when the truck accelerates and decelerates at specified rates.




***Assumptions*** **1** The acceleration remains constant. **2** Water is an incompressible substance.

***Properties*** We take the density of the water to be 1000 kg/m$^3$.

***Analysis*** (*a*) We take the *x*- and *z*- axes as shown. The horizontal acceleration is in the negative *x* direction, and thus $a_x$ is negative. Also, there is no acceleration in the vertical direction, and thus $a_z = 0$. The pressure difference between two points 1 and 2 in an incompressible fluid in linear rigid body motion is given by

$$P_2 - P_1 = -\rho a_x (x_2 - x_1) - \rho (g + a_z)(z_2 - z_1) \quad \rightarrow \quad P_2 - P_1 = -\rho a_x (x_2 - x_1)$$

since $z_2 - z_1 = 0$ along a horizontal line. Therefore, the pressure difference between the front and back of the tank is due to acceleration in the horizontal direction and the resulting compression effect towards the back of the tank. Then the pressure difference along a horizontal line becomes

$$\Delta P = P_2 - P_1 = -\rho a_x (x_2 - x_1) = -(1000 \text{ kg/m}^3)(-3 \text{ m/s}^2)(7 \text{ m})\left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right) = 21 \text{ kN/m}^2 = \mathbf{21\ kPa}$$

since $x_1 = 0$ and $x_2 = 7$ m.

(*b*) The pressure difference during deceleration is determined the way, but $a_x = 4$ m/s$^2$ in this case,

$$\Delta P = P_2 - P_1 = -\rho a_x (x_2 - x_1) = -(1000 \text{ kg/m}^3)(4 \text{ m/s}^2)(7 \text{ m})\left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right) = -28 \text{ kN/m}^2 = \mathbf{-28\ kPa}$$

***Discussion*** Note that the pressure is higher at the back end of the tank during acceleration, but at the front end during deceleration (during breaking, for example) as expected.

---

## Review Problems

### 3-110

**Solution** One section of the duct of an air-conditioning system is laid underwater. The upward force the water exerts on the duct is to be determined.

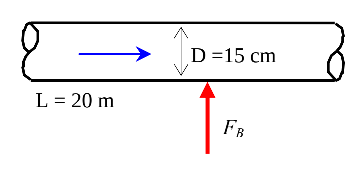
***Assumptions*** **1** The diameter given is the outer diameter of the duct (or, the thickness of the duct material is negligible). **2** The weight of the duct and the air in is negligible.

***Properties*** The density of air is given to be $\rho = 1.30$ kg/m$^3$. We take the density of water to be 1000 kg/m$^3$.

***Analysis*** Noting that the weight of the duct and the air in it is negligible, the net upward force acting on the duct is the buoyancy force exerted by water. The volume of the underground section of the duct is

$$V = AL = (\pi D^2/4)L = [\pi(0.15\text{ m})^2/4](20\text{ m}) = 0.3534\ \text{m}^3$$

Then the buoyancy force becomes




$$F_B = \rho g V = (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.3534\text{ m}^3)\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = \mathbf{3.47\ kN}$$

***Discussion*** The upward force exerted by water on the duct is 3.47 kN, which is equivalent to the weight of a mass of 354 kg. Therefore, this force must be treated seriously.

---

### 3-111

**Solution** A helium balloon tied to the ground carries 2 people. The acceleration of the balloon when it is first released is to be determined.

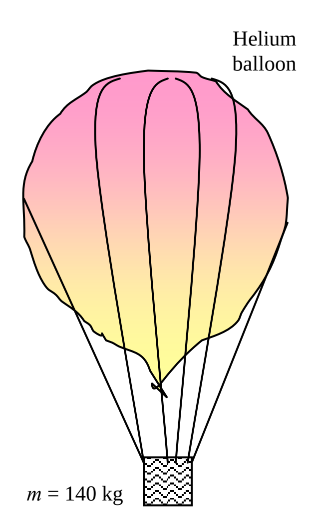
***Assumptions*** The weight of the cage and the ropes of the balloon is negligible.

***Properties*** The density of air is given to be $\rho = 1.16$ kg/m$^3$. The density of helium gas is 1/7th of this.

***Analysis*** The buoyancy force acting on the balloon is

$$V_{\text{balloon}} = 4\pi r^3/3 = 4\pi(5\ \text{m})^3/3 = 523.6\ \text{m}^3$$

$$F_B = \rho_{\text{air}} g V_{\text{balloon}} = (1.16\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(523.6\ \text{m}^3)\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = 5958.4\ \text{N}$$




The total mass is

$$m_{\text{He}} = \rho_{\text{He}} V = \left(\frac{1.16}{7}\ \text{kg/m}^3\right)(523.6\ \text{m}^3) = 86.8\ \text{kg}$$

$$m_{\text{total}} = m_{\text{He}} + m_{\text{people}} = 86.8 + 2\times 70 = 226.8\ \text{kg}$$

The total weight is

$$W = m_{\text{total}} g = (226.8\ \text{kg})(9.81\ \text{m/s}^2)\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = 2224.9\ \text{N}$$

Thus the net force acting on the balloon is

$$F_{\text{net}} = F_B - W = 5958.6 - 2224.9 = 3733.5\ \text{N}$$

Then the acceleration becomes

$$a = \frac{F_{\text{net}}}{m_{\text{total}}} = \frac{3733.5\ \text{N}}{226.8\ \text{kg}}\left(\frac{1\ \text{kg}\cdot\text{m/s}^2}{1\ \text{N}}\right) = \mathbf{16.5\ m/s^2}$$

***Discussion*** This is almost twice the acceleration of gravity – aerodynamic drag on the balloon acts quickly to slow down the acceleration.

---

**3-112** 

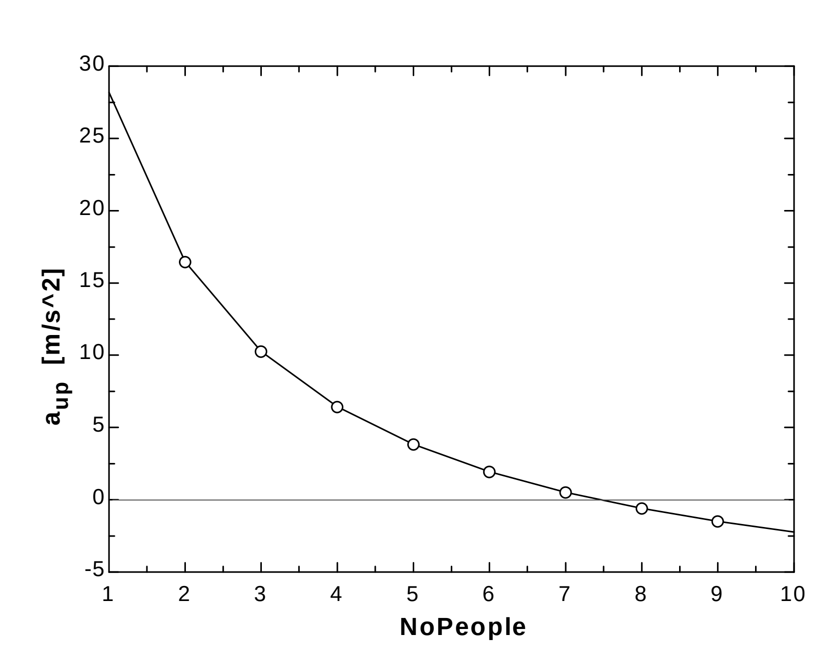
**Solution** The previous problem is reconsidered. The effect of the number of people carried in the balloon on acceleration is to be investigated. Acceleration is to be plotted against the number of people, and the results are to be discussed.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
"Given Data:"
rho_air=1.16"[kg/m^3]" "density of air"
g=9.807"[m/s^2]"
d_balloon=10"[m]"
m_1person=70"[kg]"
{NoPeople = 2}  "Data suppied in Parametric Table"

"Calculated values:"
rho_He=rho_air/7"[kg/m^3]" "density of helium"
r_balloon=d_balloon/2"[m]"
V_balloon=4*pi*r_balloon^3/3"[m^3]"
m_people=NoPeople*m_1person"[kg]"
m_He=rho_He*V_balloon"[kg]"
m_total=m_He+m_people"[kg]"
"The total weight of balloon and people is:"
W_total=m_total*g"[N]"
"The buoyancy force acting on the balloon, F_b, is equal to the weight of the air displaced by the
balloon."
F_b=rho_air*V_balloon*g"[N]"
"From the free body diagram of the balloon, the balancing vertical forces must equal the product of the
total mass and the vertical acceleration:"
F_b- W_total=m_total*a_up
```

| $A_{up}$ [m/s$^2$] | No. People |
|---|---|
| 28.19 | 1 |
| 16.46 | 2 |
| 10.26 | 3 |
| 6.434 | 4 |
| 3.831 | 5 |
| 1.947 | 6 |
| 0.5204 | 7 |
| -0.5973 | 8 |
| -1.497 | 9 |
| -2.236 | 10 |



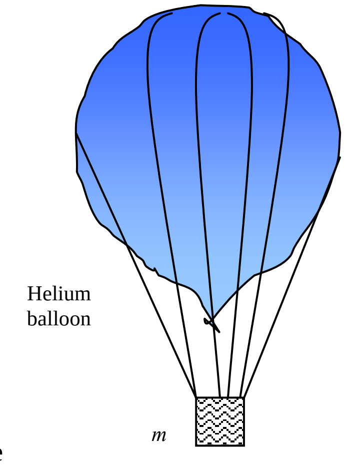

***Discussion*** As expected, the more people, the slower the acceleration. In fact, if more than 7 people are on board, the balloon does not rise at all.

**3-113**

**Solution** A balloon is filled with helium gas. The maximum amount of load the balloon can carry is to be determined.

***Assumptions*** The weight of the cage and the ropes of the balloon is negligible.

***Properties*** The density of air is given to be $\rho$ = 1.16 kg/m$^3$. The density of helium gas is 1/7th of this.




***Analysis*** In the limiting case, the net force acting on the balloon will be zero. That is, the buoyancy force and the weight will balance each other:

$$W = mg = F_B$$

$$m_{total} = \frac{F_B}{g} = \frac{5958.4 \text{ N}}{9.81 \text{ m/s}^2} = 607.4 \text{ kg}$$

Thus,

$$m_{\text{people}} = m_{\text{total}} - m_{\text{He}} = 607.4 - 86.8 = 520.6 \text{ kg} \cong \mathbf{521 \text{ kg}}$$

***Discussion*** When the net weight of the balloon and its cargo exceeds the weight of the air it displaces, the balloon/cargo is no longer "lighter than air", and therefore cannot rise.

**3-114E**

**Solution** The pressure in a steam boiler is given in kgf/cm$^2$. It is to be expressed in psi, kPa, atm, and bars.

***Analysis*** We note that 1 atm = 1.03323 kgf/cm$^2$, 1 atm = 14.696 psi, 1 atm = 101.325 kPa, and 1 atm = 1.01325 bar (inner cover page of text). Then the desired conversions become:

In atm: $$P = (75 \text{ kgf/cm}^2)\left(\frac{1 \text{ atm}}{1.03323 \text{ kgf/cm}^2}\right) = 72.6 \text{ atm}$$

In psi: $$P = (75 \text{ kgf/cm}^2)\left(\frac{1 \text{ atm}}{1.03323 \text{ kgf/cm}^2}\right)\left(\frac{14.696 \text{ psi}}{1 \text{ atm}}\right) = 1067 \text{ psi} \cong \mathbf{1070 \text{ psi}}$$

In kPa: $$P = (75 \text{ kgf/cm}^2)\left(\frac{1 \text{ atm}}{1.03323 \text{ kgf/cm}^2}\right)\left(\frac{101.325 \text{ kPa}}{1 \text{ atm}}\right) = 7355 \text{ kPa} \cong \mathbf{7360 \text{ kPa}}$$

In bars: $$P = (75 \text{ kgf/cm}^2)\left(\frac{1 \text{ atm}}{1.03323 \text{ kgf/cm}^2}\right)\left(\frac{1.01325 \text{ bar}}{1 \text{ atm}}\right) = 73.55 \text{ bar} \cong \mathbf{73.6 \text{ bar}}$$

***Discussion*** Note that the units atm, kgf/cm$^2$, and bar are almost identical to each other. All final results are given to three significant digits, but conversion ratios are typically precise to at least five significant digits.

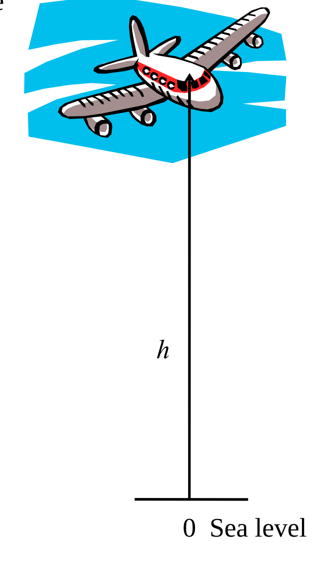
**3-115**

**Solution** A barometer is used to measure the altitude of a plane relative to the ground. The barometric readings at the ground and in the plane are given. The altitude of the plane is to be determined.

***Assumptions*** The variation of air density with altitude is negligible.

***Properties*** The densities of air and mercury are given to be $\rho_{air}$ = 1.20 kg/m$^3$ and $\rho_{mercury}$ = 13,600 kg/m$^3$.

***Analysis*** Atmospheric pressures at the location of the plane and the ground level are




$$P_{plane} = (\rho g h)_{plane}$$
$$= (13{,}600 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(0.690 \text{ m})\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ kPa}}{1000 \text{ N/m}^2}\right)$$
$$= 92.06 \text{ kPa}$$

$$P_{ground} = (\rho g h)_{ground}$$
$$= (13{,}600 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(0.753 \text{ m})\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ kPa}}{1000 \text{ N/m}^2}\right)$$
$$= 100.46 \text{ kPa}$$

Taking an air column between the airplane and the ground and writing a force balance per unit base area, we obtain

$$W_{air} / A = P_{ground} - P_{plane}$$
$$(\rho g h)_{air} = P_{ground} - P_{plane}$$
$$(1.20 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(h)\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ kPa}}{1000 \text{ N/m}^2}\right) = (100.46 - 92.06) \text{ kPa}$$

It yields $h$ = **714 m,** which is also the altitude of the airplane.

***Discussion*** Obviously, a mercury barometer is not practical on an airplane – an electronic barometer is used instead.

---

**3-116**

**Solution** A 10-m high cylindrical container is filled with equal volumes of water and oil. The pressure difference between the top and the bottom of the container is to be determined.

***Properties*** The density of water is given to be $\rho$ = 1000 kg/m$^3$. The specific gravity of oil is given to be 0.85.

***Analysis*** The density of the oil is obtained by multiplying its specific gravity by the density of water,

Oil
SG = 0.85
Water
$h$ = 10 m

$$\rho = \text{SG} \times \rho_{H_2O} = (0.85)(1000 \text{ kg/m}^3) = 850 \text{ kg/m}^3$$

The pressure difference between the top and the bottom of the cylinder is the sum of the pressure differences across the two fluids,

$$\Delta P_{total} = \Delta P_{oil} + \Delta P_{water} = (\rho g h)_{oil} + (\rho g h)_{water}$$
$$= \left[(850 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(5 \text{ m}) + (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(5 \text{ m})\right]\left(\frac{1 \text{ kPa}}{1000 \text{ N/m}^2}\right)$$
$$= \mathbf{90.7 \text{ kPa}}$$

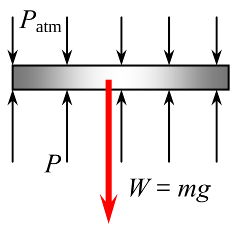
***Discussion*** The pressure at the interface must be the same in the oil and the water. Therefore, we can use the rules for hydrostatics across the two fluids, since they are at rest and there are no appreciable surface tension effects.

---

**3-117**

**Solution** The pressure of a gas contained in a vertical piston-cylinder device is measured to be 500 kPa. The mass of the piston is to be determined.

***Assumptions*** There is no friction between the piston and the cylinder.




***Analysis*** Drawing the free body diagram of the piston and balancing the vertical forces yield

$$W = PA - P_{atm}A$$

$$mg = (P - P_{atm})A$$

$$(m)(9.81\ \text{m/s}^2) = (500 - 100\ \text{kPa})(30\times10^{-4}\ \text{m}^2)\left(\frac{1000\ \text{kg/m}\cdot\text{s}^2}{1\ \text{kPa}}\right)$$

Solution of the above equation yields $m$ = **122 kg.**

***Discussion*** The gas cannot distinguish between pressure due to the piston weight and atmospheric pressure – both "feel" like a higher pressure acting on the top of the gas in the cylinder.

---

**3-118**

**Solution** The gage pressure in a pressure cooker is maintained constant at 100 kPa by a petcock. The mass of the petcock is to be determined.

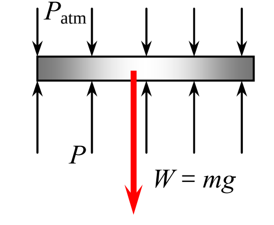
***Assumptions*** There is no blockage of the pressure release valve.

***Analysis*** Atmospheric pressure is acting on all surfaces of the petcock, which balances itself out. Therefore, it can be disregarded in calculations if we use the gage pressure as the cooker pressure. A force balance on the petcock ($\Sigma F_y = 0$) yields

$$W = P_{gage}A$$

$$m = \frac{P_{gage}A}{g} = \frac{(100\ \text{kPa})(4\times10^{-6}\ \text{m}^2)}{9.81\ \text{m/s}^2}\left(\frac{1000\ \text{kg/m}\cdot\text{s}^2}{1\ \text{kPa}}\right)$$

$$= \mathbf{0.0408\ kg = 40.8\ g}$$




***Discussion*** The higher pressure causes water in the cooker to boil at a higher temperature.

---

**3-119**

**Solution** A glass tube open to the atmosphere is attached to a water pipe, and the pressure at the bottom of the tube is measured. It is to be determined how high the water will rise in the tube.

***Properties*** The density of water is given to be $\rho = 1000\ \text{kg/m}^3$.

***Analysis*** The pressure at the bottom of the tube can be expressed as

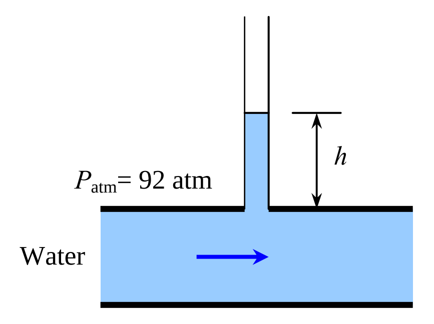
$$P = P_{atm} + (\rho g h)_{tube}$$

Solving for $h$,

$$h = \frac{P - P_{atm}}{\rho g}$$

$$= \frac{(115-92)\ \text{kPa}}{(1000\ \text{kg/m}^3)(9.8\ \text{m/s}^2)}\left(\frac{1\ \text{kg}\cdot\text{m/s}^2}{1\ \text{N}}\right)\left(\frac{1000\ \text{N/m}^2}{1\ \text{kPa}}\right)$$

$$= \mathbf{2.35\ m}$$




***Discussion*** Even though the water is flowing, the water in the tube itself is at rest. If the pressure at the tube bottom had been given in terms of gage pressure, we would not have had to take into account the atmospheric pressure term.

---

### 3-120

**Solution** The average atmospheric pressure is given as $P_{\text{atm}} = 101.325(1 - 0.02256\,z)^{5.256}$ where $z$ is the altitude in km. The atmospheric pressures at various locations are to be determined.

***Analysis*** Atmospheric pressure at various locations is obtained by substituting the altitude z values in km into the relation $P_{atm} = 101.325(1 - 0.02256z)^{5.256}$. The results are tabulated below.

| | |
|---|---|
| Atlanta: | (z = 0.306 km): $P_{atm}$ = 101.325(1 - 0.02256×0.306)$^{5.256}$ = **97.7 kPa** |
| Denver: | (z = 1.610 km): $P_{atm}$ = 101.325(1 - 0.02256×1.610)$^{5.256}$ = **83.4 kPa** |
| M. City: | (z = 2.309 km): $P_{atm}$ = 101.325(1 - 0.02256×2.309)$^{5.256}$ = **76.5 kPa** |
| Mt. Ev.: | (z = 8.848 km): $P_{atm}$ = 101.325(1 - 0.02256×8.848)$^{5.256}$ = **31.4 kPa** |

***Discussion*** It may be surprising, but the atmospheric pressure on Mt. Everest is less than 1/3 that at sea level!

---

### 3-121

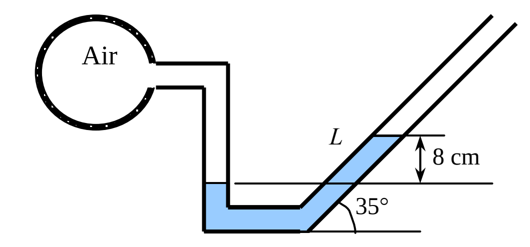
**Solution** The air pressure in a duct is measured by an inclined manometer. For a given vertical level difference, the gage pressure in the duct and the length of the differential fluid column are to be determined.

***Assumptions*** The manometer fluid is an incompressible substance.

***Properties*** The density of the liquid is given to be $\rho$ = 0.81 kg/L = 810 kg/m$^3$.

***Analysis*** The gage pressure in the duct is determined from

$$P_{\text{gage}} = P_{\text{abs}} - P_{\text{atm}} = \rho g h$$
$$= (810\,\text{kg/m}^3)(9.81\ \text{m/s}^2)(0.08\,\text{m})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{Pa}}{1\,\text{N/m}^2}\right)$$
$$= \mathbf{636\ \ Pa}$$




The length of the differential fluid column is

$$L = h/\sin\theta = (8\ \text{cm})/\sin 35° = \mathbf{13.9\ \ cm}$$

***Discussion*** Note that the length of the differential fluid column is extended considerably by inclining the manometer arm for better readability (and therefore higher *precision*).

---

### 3-122E

**Solution** Equal volumes of water and oil are poured into a U-tube from different arms, and the oil side is pressurized until the contact surface of the two fluids moves to the bottom and the liquid levels in both arms become the same. The excess pressure applied on the oil side is to be determined.

***Assumptions*** **1** Both water and oil are incompressible substances. **2** Oil does not mix with water. **3** The cross-sectional area of the U-tube is constant.

***Properties*** The density of oil is given to be $\rho_{\text{oil}}$ = 49.3 lbm/ft$^3$. We take the density of water to be $\rho_{\text{w}}$ = 62.4 lbm/ft$^3$.

***Analysis*** Noting that the pressure of both the water and the oil is the same at the contact surface, the pressure at this surface can be expressed as

$$P_{\text{contact}} = P_{\text{blow}} + \rho_{\text{a}} g h_{\text{a}} = P_{\text{atm}} + \rho_{\text{w}} g h_{\text{w}}$$

Water
Blown air
Oil
30 in

Noting that $h_a = h_w$ and rearranging,

$$P_{\text{gage, blow}} = P_{\text{blow}} - P_{\text{atm}} = (\rho_w - \rho_{\text{oil}})gh$$
$$= (62.4\text{- }49.3\ \ \text{lbm/ft}^3)(32.2\ \text{ft/s}^2)(30/12\ \text{ft})\left(\frac{1\ \ \text{lbf}}{32.2\ \ \text{lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1\ \ \text{ft}^2}{144\ \ \text{in}^2}\right)$$
$$= \mathbf{0.227\ \ psi}$$

***Discussion*** When the person stops blowing, the oil rises and some water flows into the right arm. It can be shown that when the curvature effects of the tube are disregarded, the differential height of water is 23.7 in to balance 30-in of oil.

---

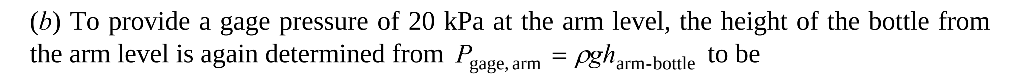
**3-123**

**Solution** It is given that an IV fluid and the blood pressures balance each other when the bottle is at a certain height, and a certain gage pressure at the arm level is needed for sufficient flow rate. The gage pressure of the blood and elevation of the bottle required to maintain flow at the desired rate are to be determined.

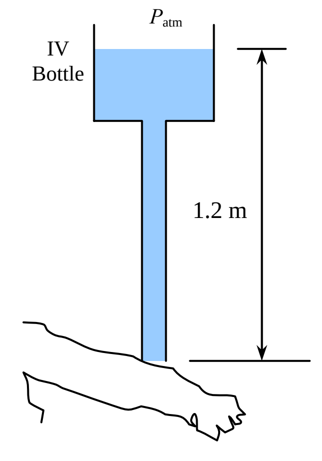
***Assumptions*** **1** The IV fluid is incompressible. **2** The IV bottle is open to the atmosphere.

***Properties*** The density of the IV fluid is given to be $\rho = 1020$ kg/m$^3$.

***Analysis*** (*a*) Noting that the IV fluid and the blood pressures balance each other when the bottle is 1.2 m above the arm level, the gage pressure of the blood in the arm is simply equal to the gage pressure of the IV fluid at a depth of 1.2 m,



$$P_{\text{gage, arm}} = P_{\text{abs}} - P_{\text{atm}} = \rho g h_{\text{arm-bottle}}$$
$$= (1020 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(1.20 \text{ m})\left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ kPa}}{1 \text{ kN/m}^2}\right)$$
$$= \mathbf{12.0 \text{ kPa}}$$




(*b*) To provide a gage pressure of 20 kPa at the arm level, the height of the bottle from the arm level is again determined from $P_{\text{gage, arm}} = \rho g h_{\text{arm-bottle}}$ to be

$$h_{\text{arm-bottle}} = \frac{P_{\text{gage, arm}}}{\rho g}$$
$$= \frac{20 \text{ kPa}}{(1020 \text{ kg/m}^3)(9.81 \text{ m/s}^2)}\left(\frac{1000 \text{ kg}\cdot\text{m/s}^2}{1 \text{ kN}}\right)\left(\frac{1 \text{ kN/m}^2}{1 \text{ kPa}}\right) = \mathbf{2.0 \text{ m}}$$

***Discussion*** Note that the height of the reservoir can be used to control flow rates in gravity driven flows. When there is flow, the pressure drop in the tube due to friction should also be considered. This will result in raising the bottle a little higher to overcome pressure drop.

**3-124**

**Solution** A gasoline line is connected to a pressure gage through a double-U manometer. For a given reading of the pressure gage, the gage pressure of the gasoline line is to be determined.

***Assumptions*** **1** All the liquids are incompressible. **2** The effect of air column on pressure is negligible.

***Properties*** The specific gravities of oil, mercury, and gasoline are given to be 0.79, 13.6, and 0.70, respectively. We take the density of water to be $\rho_w$ = 1000 kg/m$^3$.

***Analysis*** Starting with the pressure indicated by the pressure gage and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the gasoline pipe, and setting the result equal to $P_{\text{gasoline}}$ gives

$$P_{\text{gage}} - \rho_{\text{w}} g h_w + \rho_{\text{oil}} g h_{\text{oil}} - \rho_{\text{Hg}} g h_{\text{Hg}} - \rho_{\text{gasoline}} g h_{\text{gasoline}} = P_{\text{gasoline}}$$

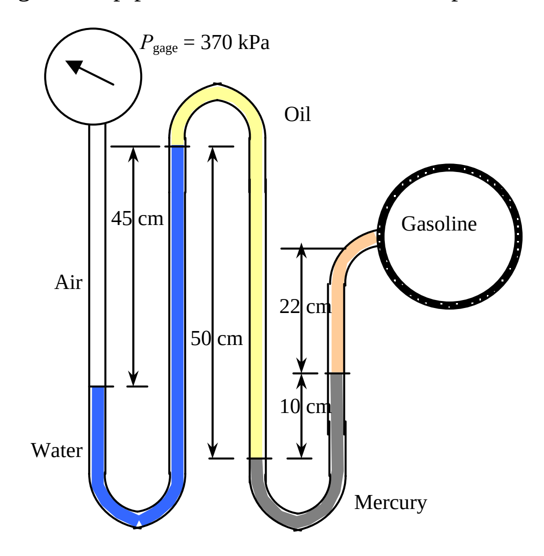
Rearranging,

$$P_{\text{gasoline}} = P_{\text{gage}} - \rho_{\text{w}} g(h_w - SG_{\text{oil}} h_{\text{oil}} + SG_{\text{Hg}} h_{\text{Hg}} + SG_{\text{gasoline}} h_{\text{gasoline}})$$

Substituting,

$$P_{\text{gasoline}} = 370 \text{ kPa -(1000 kg/m}^3\text{)(9.81 m/s}^2\text{)[(0.45 m)} - 0.79(0.5\text{ m}) + 13.6(0.1\text{ m}) + 0.70(0.22\text{ m})]$$
$$\times \left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right)$$
$$= 354.6\text{ kPa} \cong \mathbf{355\ kPa}$$

Therefore, the pressure in the gasoline pipe is 15.4 kPa lower than the pressure reading of the pressure gage.




***Discussion*** Note that sometimes the use of specific gravity offers great convenience in the solution of problems that involve several fluids.

**3-125**

**Solution** A gasoline line is connected to a pressure gage through a double-U manometer. For a given reading of the pressure gage, the gage pressure of the gasoline line is to be determined.

***Assumptions*** **1** All the liquids are incompressible. **2** The effect of air column on pressure is negligible.

***Properties*** The specific gravities of oil, mercury, and gasoline are given to be 0.79, 13.6, and 0.70, respectively. We take the density of water to be $\rho_w$ = 1000 kg/m$^3$.

***Analysis*** Starting with the pressure indicated by the pressure gage and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the gasoline pipe, and setting the result equal to $P_{\text{gasoline}}$ gives

$$P_{gage} - \rho_{\text{w}} g h_w + \rho_{\text{alcohol}} g h_{\text{alcohol}} - \rho_{\text{Hg}} g h_{\text{Hg}} - \rho_{\text{gasoline}} g h_{\text{gasoline}} = P_{\text{gasoline}}$$

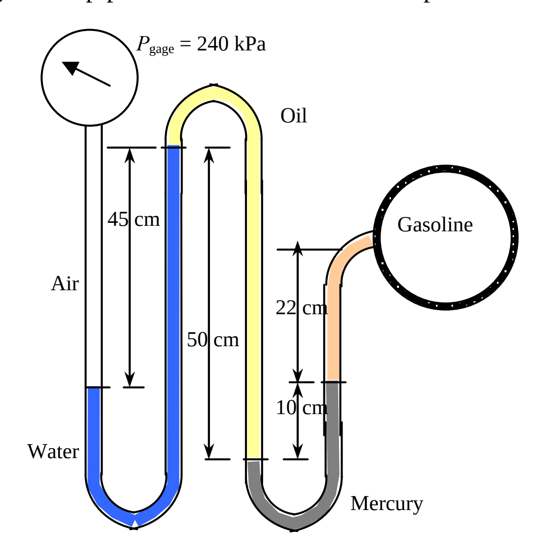
Rearranging,

$$P_{\text{gasoline}} = P_{\text{gage}} - \rho_{\text{w}} g (h_w - SG_{\text{alcohol}} h_{\text{s,alcohol}} + SG_{\text{Hg}} h_{\text{Hg}} + SG_{\text{gasoline}} h_{\text{s,gasoline}})$$

Substituting,

$$P_{\text{gasoline}} = 240 \text{ kPa} - (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)[(0.45 \text{ m}) - 0.79(0.5 \text{ m}) + 13.6(0.1 \text{ m}) + 0.70(0.22 \text{ m})]$$
$$\times \left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ kPa}}{1 \text{ kN/m}^2}\right)$$
$$= 224.6 \text{ kPa} \cong \mathbf{225 \text{ kPa}}$$

Therefore, the pressure in the gasoline pipe is 15.4 kPa lower than the pressure reading of the pressure gage.




***Discussion*** Note that sometimes the use of specific gravity offers great convenience in the solution of problems that involve several fluids.

**3-126E**

**Solution** A water pipe is connected to a double-U manometer whose free arm is open to the atmosphere. The absolute pressure at the center of the pipe is to be determined.

***Assumptions*** **1** All the liquids are incompressible. **2** The solubility of the liquids in each other is negligible.

***Properties*** The specific gravities of mercury and oil are given to be 13.6 and 0.80, respectively. We take the density of water to be $\rho_w$ = 62.4 lbm/ft$^3$.

***Analysis*** Starting with the pressure at the center of the water pipe, and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the free surface of oil where the oil tube is exposed to the atmosphere, and setting the result equal to $P_{atm}$ gives

$$P_{\text{water pipe}} - \rho_{\text{water}} g h_{\text{water}} + \rho_{\text{alcohol}} g h_{\text{alcohol}} - \rho_{\text{Hg}} g h_{\text{Hg}} - \rho_{\text{oil}} g h_{\text{oil}} = P_{atm}$$

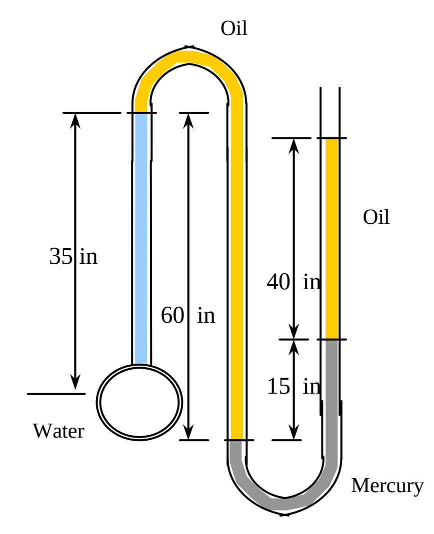
Solving for $P_{\text{water pipe}}$,

$$P_{\text{water pipe}} = P_{atm} + \rho_{\text{water}} g (h_{\text{water}} - SG_{\text{oil}} h_{\text{alcohol}} + SG_{\text{Hg}} h_{\text{Hg}} + SG_{\text{oil}} h_{\text{oil}})$$

Substituting,

$$P_{\text{water pipe}} = 14.2\,\text{psia} + (62.4\,\text{lbm/ft}^3)(32.2\,\text{ft/s}^2)[(35/12\,\text{ft}) - 0.80(60/12\,\text{ ft}) + 13.6(15/12\,\text{ft})$$
$$+ 0.8(40/12\,\text{ft})] \times \left(\frac{1\,\text{lbf}}{32.2\,\text{lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1\,\text{ft}^2}{144\,\text{in}^2}\right)$$
$$= \mathbf{22.3\ psia}$$

Therefore, the absolute pressure in the water pipe is 22.3 psia.




***Discussion*** Note that jumping horizontally from one tube to the next and realizing that pressure remains the same in the same fluid simplifies the analysis greatly.

**3-127**

**Solution** The pressure of water flowing through a pipe is measured by an arrangement that involves both a pressure gage and a manometer. For the values given, the pressure in the pipe is to be determined.

***Assumptions*** **1** All the liquids are incompressible. **2** The effect of air column on pressure is negligible.

***Properties*** The specific gravity of gage fluid is given to be 2.4. We take the standard density of water to be $\rho_w$ = 1000 kg/m$^3$.

***Analysis*** Starting with the pressure indicated by the pressure gage and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the water pipe, and setting the result equal to $P_{\text{water}}$ give

$$P_{\text{gage}} + \rho_{\text{w}} g h_{w1} - \rho_{\text{gage}} g h_{\text{gage}} - \rho_{\text{w}} g h_{\text{w2}} = P_{\text{water}}$$

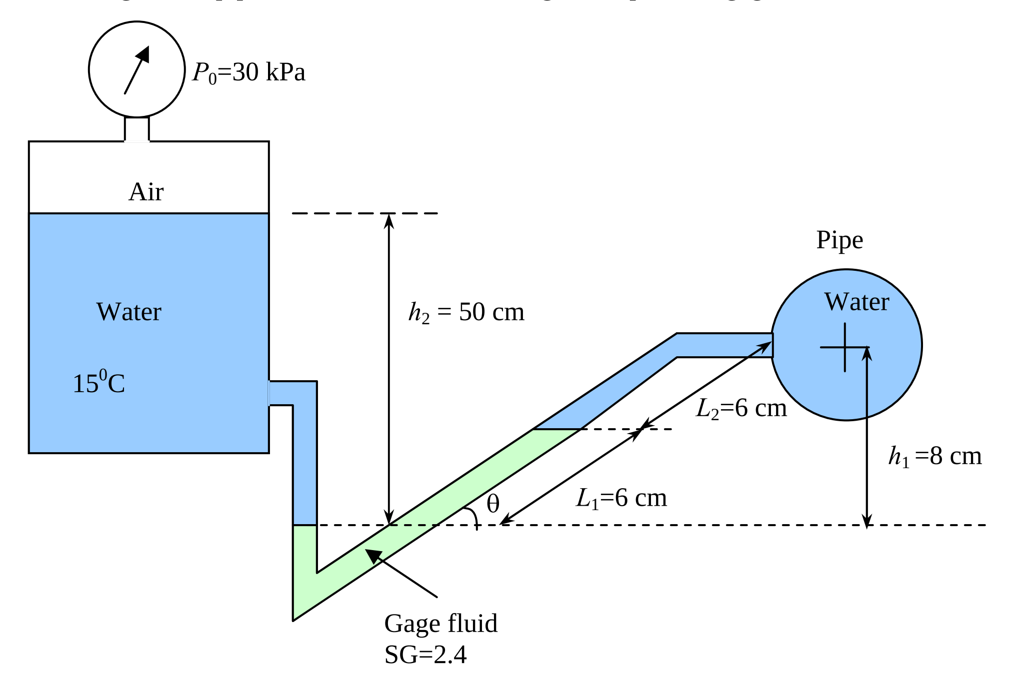
Rearranging,

$$P_{\text{water}} = P_{\text{gage}} + \rho_{\text{w}} g\left(h_{w1} - \text{SG}_{\text{gage}} h_{\text{gage}} - h_{w2}\right) = P_{\text{gage}} + \rho_{\text{w}} g\left(h_2 - \text{SG}_{\text{gage}} L_1 \sin\theta - L_2 \sin\theta\right)$$

Noting that $\sin\theta = 8/12 = 0.6667$ and substituting,

$$P_{\text{water}} = 30\text{ kPa} + (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)[(0.50\text{ m}) - 2.4(0.06\text{ m})0.6667 - (0.06\text{ m})0.6667]$$
$$\times\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right)$$
$$= \mathbf{33.6\ kPa}$$

Therefore, the pressure in the gasoline pipe is 3.6 kPa over the reading of the pressure gage.




***Discussion*** Note that even without a manometer, the reading of a pressure gage can be in error if it is not placed at the same level as the pipe when the fluid is a liquid.

**3-128**

**Solution** A U-tube filled with mercury except the 18-cm high portion at the top. Oil is poured into the left arm, forcing some mercury from the left arm into the right one. The maximum amount of oil that can be added into the left arm is to be determined.

***Assumptions*** **1** Both liquids are incompressible. **2** The U-tube is perfectly vertical.

***Properties*** The specific gravities are given to be 2.72 for oil and 13.6 for mercury.

***Analysis*** Initially, the mercury levels in both tubes are the same. When oil is poured into the left arm, it will push the mercury in the left down, which will cause the mercury level in the right arm to rise. Noting that the volume of mercury is constant, the decrease in the mercury volume in left column must be equal to the increase in the mercury volume in the right arm. Therefore, if the drop in mercury level in the left arm is *x*, the rise in the mercury level in the right arm *h* corresponding to a drop of *x* in the left arm is

$$V_{\text{left}} = V_{\text{right}} \quad \rightarrow \quad \pi(2d)^2 x = \pi d^2 h \quad \rightarrow \quad h = 4x$$

The pressures at points *A* and *B* are equal $P_A = P_B$ and thus

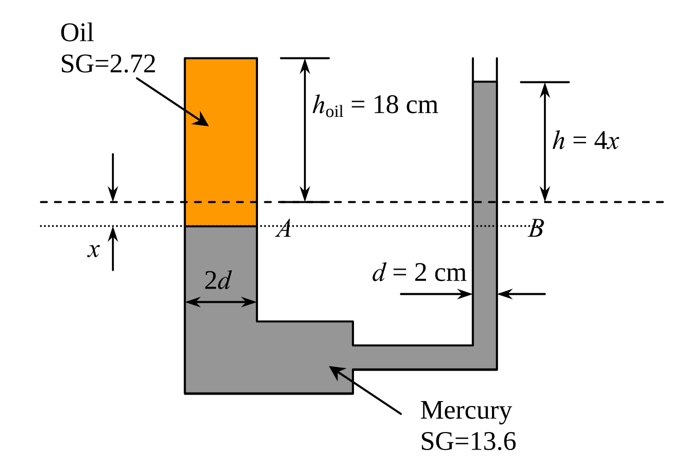
$$P_{atm} + \rho_{\text{oil}} g(h_{oil} + x) = P_{atm} + \rho_{Hg} g h_{Hg} \qquad \rightarrow \quad \text{SG}_{\text{oil}} \rho_w g\left(h_{\text{oil}} + x\right) = \text{SG}_{\text{Hg}} \rho_w g\left(5x\right)$$

Solving for *x* and substituting,

$$x = \frac{\text{SG}_{\text{oil}} h_{\text{oil}}}{5\text{SG}_{\text{Hg}} - \text{SG}_{\text{oil}}} = \frac{2.72\left(18\text{ cm}\right)}{5 \times 13.6 - 2.72} = 0.75\text{ cm}$$

Therefore, the maximum amount of oil that can be added into the left arm is

$$V_{\text{oil, max}} = \pi(2d/2)^2 (h_{\text{oil}} + x) = \pi(2\text{ cm})^2 (18 + 0.75\text{ cm}) = \mathbf{236\ cm^3} = \mathbf{0.236\ L}$$




***Discussion*** Note that the fluid levels in the two arms of a U-tube can be different when two different fluids are involved.

**3-129**

**Solution** The pressure buildup in a teapot may cause the water to overflow through the service tube. The maximum cold-water height to avoid overflow under a specified gage pressure is to be determined.

***Assumptions*** **1** Water is incompressible. **2** Thermal expansion and the amount of water in the service tube are negligible. **3** The cold water temperature is 20°C.

***Properties*** The density of water at 20°C is $\rho_w$ = 998.0 kg/m$^3$.

***Analysis*** From geometric considerations, the vertical distance between the bottom of the teapot and the tip of the service tube is

$$h_{\text{tip}} = 4 + 12\cos 40° = 13.2 \text{ cm}$$

This would be the maximum water height if there were no pressure build-up inside by the steam. The steam pressure inside the teapot above the atmospheric pressure must be balanced by the water column inside the service tube,

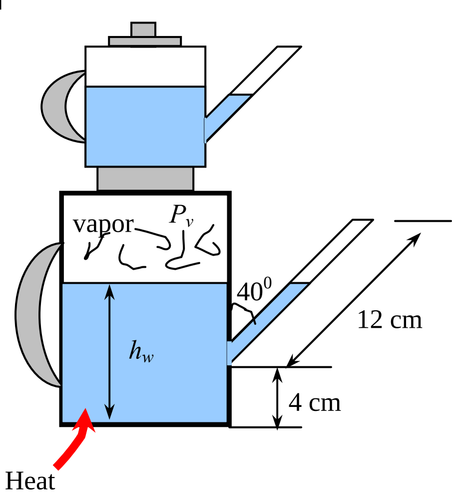
$$P_{\text{v, gage}} = \rho_w g \Delta h_w$$

or,

$$\Delta h_w = \frac{P_{\text{v, gage}}}{\rho_w g} = \frac{0.32 \text{ kPa}}{(998.0 \text{ kg/m}^3)(9.81 \text{ m/s}^2)}\left(\frac{1000 \text{ kg}\cdot\text{m/s}^2}{1 \text{ kN}}\right)\left(\frac{1 \text{ kN/m}^2}{1 \text{ kPa}}\right) = 0.033 \text{ m} = 3.3 \text{ cm}$$

Therefore, the water level inside the teapot must be 3.3 cm below the tip of the service tube. Then the maximum initial water height inside the teapot to avoid overflow becomes

$$h_{\text{w, max}} = h_{\text{tip}} - \Delta h_w = 13.2 - 3.3 = \mathbf{9.9 \text{ cm}}$$




***Discussion*** We can obtain the same result formally by starting with the vapor pressure in the teapot and moving along the service tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the atmosphere, and setting the result equal to $P_{\text{atm}}$:

$$P_{atm} + P_{v,gage} - \rho_w g h_w = P_{atm} \quad \rightarrow \quad P_{v,gage} = \rho_w g h_w$$

**3-130**

**Solution** The pressure buildup in a teapot may cause the water to overflow through the service tube. The maximum cold-water height to avoid overflow under a specified gage pressure is to be determined by considering the effect of thermal expansion.

***Assumptions*** **1** The amount of water in the service tube is negligible. **3** The cold water temperature is 20°C.

***Properties*** The density of water is $\rho_w$ = 998.0 kg/m$^3$ at 20°C, and $\rho_w$ = 957.9 kg/m$^3$ at 100°C

***Analysis*** From geometric considerations, the vertical distance between the bottom of the teapot and the tip of the service tube is

$$h_{\text{tip}} = 4 + 12\cos 40° = 13.2 \text{ cm}$$

This would be the maximum water height if there were no pressure build-up inside by the steam. The steam pressure inside the teapot above the atmospheric pressure must be balanced by the water column inside the service tube,

$$P_{\text{v, gage}} = \rho_w g \Delta h_w$$

or,

$$\Delta h_{\text{w}} = \frac{P_{\text{v, gage}}}{\rho_{\text{w}} g} = \frac{0.32 \text{ kPa}}{(998.0 \text{ kg/m}^3)(9.81 \text{ m/s}^2)}\left(\frac{1000 \text{ kg}\cdot\text{m/s}^2}{1 \text{ kN}}\right)\left(\frac{1 \text{ kN/m}^2}{1 \text{ kPa}}\right) = 0.033 \text{ m} = 3.3 \text{ cm}$$

Therefore, the water level inside the teapot must be 3.4 cm below the tip of the service tube. Then the height of hot water inside the teapot to avoid overflow becomes

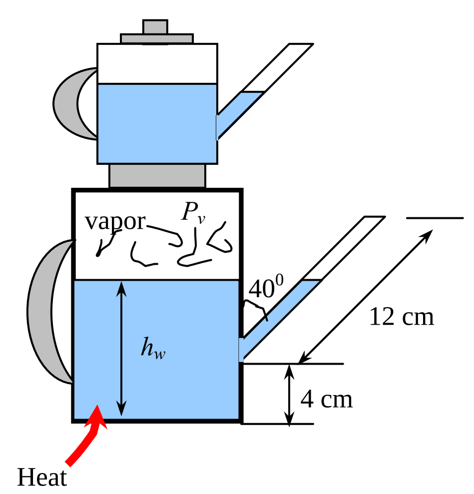
$$h_{\text{w}} = h_{\text{tip}} - \Delta h_w = 13.2 - 3.4 = 9.8 \text{ cm}$$

The specific volume of water is 1/998 m$^3$/kg at 20°C and 1/957.9 m$^3$/kg at 100°C. Then the percent drop in the volume of water as it cools from 100°C to 20°C is

$$\text{Volume reduction} = \frac{v_{100°C} - v_{20°C}}{v_{100°C}} = \frac{1/957.9 - 1/998.0}{1/957.9} = 0.040 \quad \text{or } 4.0\%$$

Volume is proportional to water height, and to allow for thermal expansion, the volume of cold water should be 4% less. Therefore, the maximum initial water height to avoid overflow should be

$$h_{\text{w, max}} = (1 - 0.040)h_{\text{w}} = 0.96 \times 9.8 \text{ cm} = \mathbf{9.4 \text{ cm}}$$




***Discussion*** Note that the effect of thermal expansion can be quite significant.

**3-131**

**Solution** The temperature of the atmosphere varies with altitude $z$ as $T = T_0 - \beta z$, while the gravitational acceleration varies by $g(z) = g_0/(1 + z/6{,}370{,}320)^2$. Relations for the variation of pressure in atmosphere are to be obtained (*a*) by ignoring and (*b*) by considering the variation of $g$ with altitude.

***Assumptions*** The air in the troposphere behaves as an ideal gas.

***Analysis*** (*a*) Pressure change across a differential fluid layer of thickness $dz$ in the vertical $z$ direction is

$$dP = -\rho g dz$$

From the ideal gas relation, the air density can be expressed as $\rho = \frac{P}{RT} = \frac{P}{R(T_0 - \beta z)}$. Then,

$$dP = -\frac{P}{R(T_0 - \beta z)} g dz$$

Separating variables and integrating from $z = 0$ where $P = P_0$ to $z = z$ where $P = P$,

$$\int_{P_0}^{P} \frac{dP}{P} = -\int_0^z \frac{g dz}{R(T_0 - \beta z)}$$

Performing the integrations.

$$\ln \frac{P}{P_0} = \frac{g}{R\beta} \ln \frac{T_0 - \beta z}{T_0}$$

Rearranging, the desired relation for atmospheric pressure for the case of constant $g$ becomes

$$\boxed{P = P_0 \left(1 - \frac{\beta z}{T_0}\right)^{\frac{g}{\beta R}}}$$

(*b*) When the variation of $g$ with altitude is considered, the procedure remains the same but the expressions become more complicated,

$$dP = -\frac{P}{R(T_0 - \beta z)} \frac{g_0}{(1 + z/6{,}370{,}320)^2} dz$$

Separating variables and integrating from $z = 0$ where $P = P_0$ to $z = z$ where $P = P$,

$$\int_{P_0}^{P} \frac{dP}{P} = -\int_0^z \frac{g_0 dz}{R(T_0 - \beta z)(1 + z/6{,}370{,}320)^2}$$

Performing the integrations,

$$\ln P \Big|_{P_0}^{P} = \frac{g_0}{R\beta} \left| \frac{1}{(1 + kT_0/\beta)(1 + kz)} - \frac{1}{(1 + kT_0/\beta)^2} \ln \frac{1 + kz}{T_0 - \beta z} \right|_0^z$$

where $R$ = 287 J/kg·K = 287 m$^2$/s$^2$·K is the gas constant of air. After some manipulations, we obtain

$$\boxed{P = P_0 \exp\left[-\frac{g_0}{R(\beta + kT_0)}\left(\frac{1}{1 + 1/kz} + \frac{1}{1 + kT_0/\beta} \ln \frac{1 + kz}{1 - \beta z/T_0}\right)\right]}$$

where $T_0$ = 288.15 K, β = 0.0065 K/m, $g_0$ = 9.807 m/s$^2$, $k$ = 1/6,370,320 m$^{-1}$, and $z$ is the elevation in m..

***Discussion*** When performing the integration in part (*b*), the following expression from integral tables is used, together with a transformation of variable $x = T_0 - \beta z$,

$$\int \frac{dx}{x(a + bx)^2} = \frac{1}{a(a + bx)} - \frac{1}{a^2} \ln \frac{a + bx}{x}$$

Also, for $z$ = 11,000 m, for example, the relations in (*a*) and (*b*) give 22.62 and 22.69 kPa, respectively.

**3-132**

**Solution** The variation of pressure with density in a thick gas layer is given. A relation is to be obtained for pressure as a function of elevation *z*.

***Assumptions*** The property relation $P = C\rho^n$ is valid over the entire region considered.

***Analysis*** The pressure change across a differential fluid layer of thickness *dz* in the vertical *z* direction is given as,

$$dP = -\rho g dz$$

Also, the relation $P = C\rho^n$ can be expressed as $C = P/\rho^n = P_0/\rho_0^n$, and thus

$$\rho = \rho_0 (P/P_0)^{1/n}$$

Substituting,

$$dP = -g\rho_0 (P/P_0)^{1/n}\, dz$$

Separating variables and integrating from $z = 0$ where $P = P_0 = C\rho_0^n$ to $z = z$ where $P = P$,

$$\int_{P_0}^{P} (P/P_0)^{-1/n} dP = -\rho_0 g \int_0^z dz$$

Performing the integrations.

$$P_0 \frac{(P/P_0)^{-1/n+1}}{-1/n+1}\Bigg|_{P_0}^{P} = -\rho_0 gz \quad \rightarrow \quad \left(\frac{P}{P_0}\right)^{(n-1)/n} - 1 = -\frac{n-1}{n}\frac{\rho_0 gz}{P_0}$$

Solving for *P*,

$$\boxed{P = P_0\left(1 - \frac{n-1}{n}\frac{\rho_0 gz}{P_0}\right)^{n/(n-1)}}$$

which is the desired relation.

***Discussion*** The final result could be expressed in various forms. The form given is very convenient for calculations as it facilitates unit cancellations and reduces the chance of error.

**3-133**

**Solution** A pressure transducer is used to measure pressure by generating analogue signals, and it is to be calibrated by measuring both the pressure and the electric current simultaneously for various settings, and the results are to be tabulated. A calibration curve in the form of $P = aI + b$ is to be obtained, and the pressure corresponding to a signal of 10 mA is to be calculated.

***Assumptions*** Mercury is an incompressible liquid.

***Properties*** The specific gravity of mercury is given to be 13.56, and thus its density is 13,560 kg/m$^3$.

***Analysis*** For a given differential height, the pressure can be calculated from

$$P = \rho g \Delta h$$

For $\Delta h$ = 28.0 mm = 0.0280 m, for example,

$$P = 13.56(1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(0.0280\ \text{m})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1\ \text{kN/m}^2}\right) = 3.72\ \text{kPa}$$

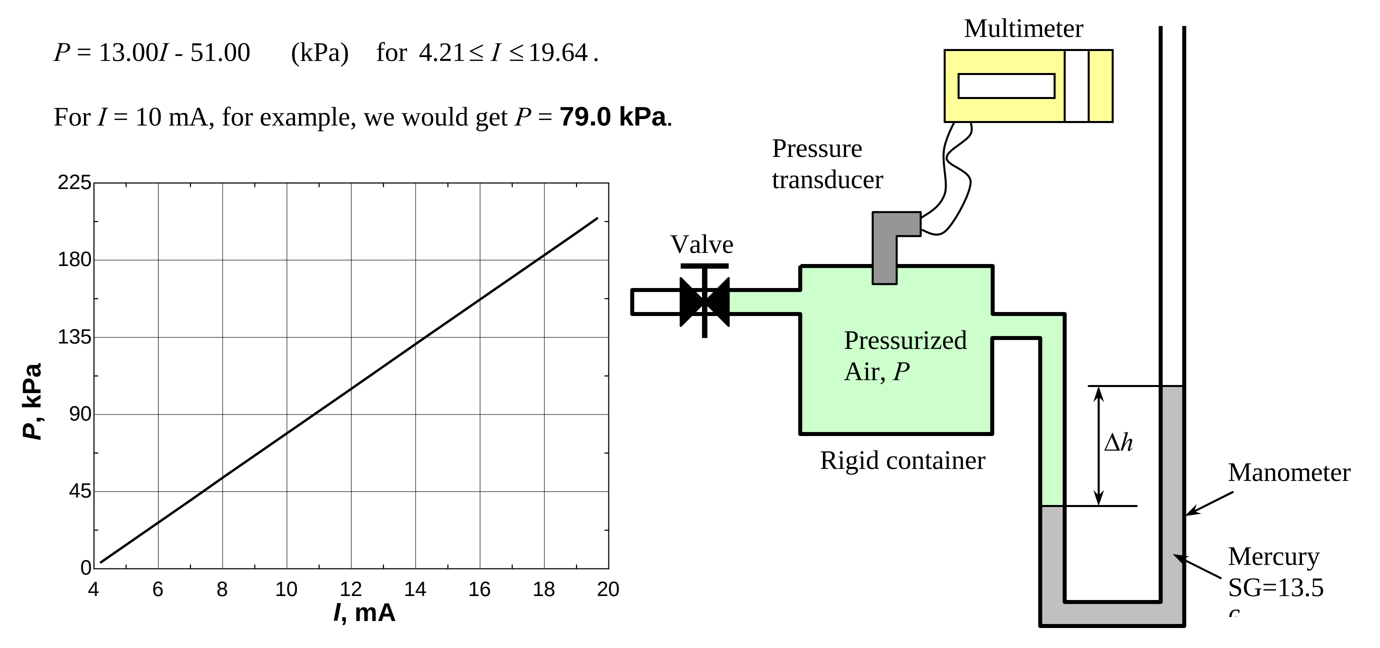
Repeating the calculations and tabulating, we have

| $\Delta h$(mm) | 28.0 | 181.5 | 297.8 | 413.1 | 765.9 | 1027 | 1149 | 1362 | 1458 | 1536 |
|---|---|---|---|---|---|---|---|---|---|---|
| ***P*(kPa)** | **3.72** | **24.14** | **39.61** | **54.95** | **101.9** | **136.6** | **152.8** | **181.2** | **193.9** | **204.3** |
| $I$ (mA) | 4.21 | 5.78 | 6.97 | 8.15 | 11.76 | 14.43 | 15.68 | 17.86 | 18.84 | 19.64 |

A plot of $P$ versus $I$ is given below. It is clear that the pressure varies linearly with the current, and using EES, the best curve fit is obtained to be

$P = 13.00I - 51.00$ (kPa) for $4.21 \le I \le 19.64$.

For $I$ = 10 mA, for example, we would get $P$ = **79.0 kPa**.




***Discussion*** Note that the calibration relation is valid in the specified range of currents or pressures.

**3-134**

**Solution** A system is equipped with two pressure gages and a manometer. For a given differential fluid height, the pressure difference $\Delta P = P_2 - P_1$ is to be determined.

***Assumptions*** **1** All the liquids are incompressible. **2** The effect of air column on pressure is negligible.

***Properties*** The specific gravities are given tone 2.67 for the gage fluid and 0.87 for oil. We take the standard density of water to be $\rho_w$ = 1000 kg/m$^3$.

***Analysis*** Starting with the pressure indicated by the pressure gage 2 and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms and ignoring the air spaces until we reach the pressure gage 1, and setting the result equal to $P_1$ give

$$P_2 - \rho_{\text{gage}} g h_{\text{gage}} + \rho_{\text{oil}} g h_{\text{oil}} = P_1$$

Rearranging,

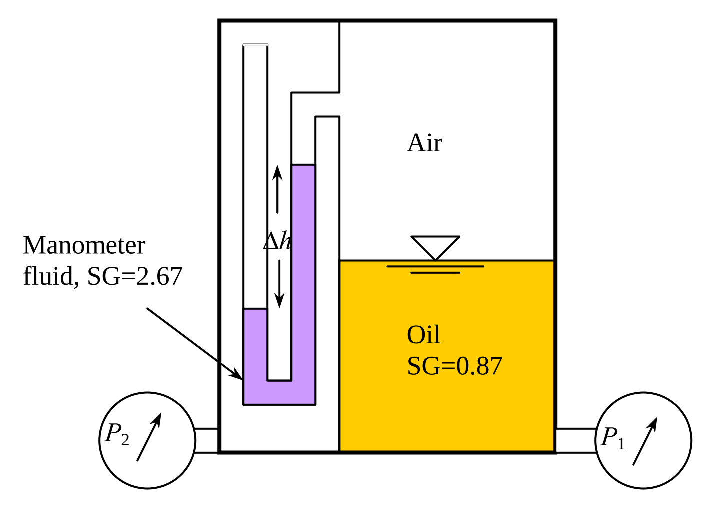
$$P_2 - P_1 = \rho_w g \left(\text{SG}_{\text{gage}} h_{\text{gage}} - \text{SG}_{\text{oil}} h_{\text{oil}}\right)$$

Substituting,

$$P_2 - P_1 = (1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)[2.67(0.08\ \text{m}) - 0.87(0.65\ \text{m})]\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1\ \text{kN/m}^2}\right)$$

$$= \mathbf{-3.45\ kPa}$$

Therefore, the pressure reading of the left gage is 3.45 kPa lower than that of the right gage.




***Discussion*** The negative pressure difference indicates that the pressure differential across the oil level is greater than the pressure differential corresponding to the differential height of the manometer fluid.

**3-135**

**Solution** An oil pipeline and a rigid air tank are connected to each other by a manometer. The pressure in the pipeline and the change in the level of manometer fluid due to a air temperature drop are to be determined.

***Assumptions*** **1** All the liquids are incompressible. **2** The effect of air column on pressure is negligible. **3** The air volume in the manometer is negligible compared with the volume of the tank.

***Properties*** The specific gravities are given to be 2.68 for oil and 13.6 for mercury. We take the standard density of water to be $\rho_w$ = 1000 kg/m$^3$. The gas constant of air is 0.287 kPa·m$^3$/kg.K.

***Analysis*** (*a*) Starting with the oil pipe and moving along the tube by adding (as we go down) or subtracting (as we go up) the $\rho gh$ terms until we reach the air tank, and setting the result equal to $P_{air}$ give

$$P_{oil} + \rho_{oil} g h_{oil} + \rho_{Hg} g h_{Hg} = P_{air}$$

The absolute pressure in the air tank is determined from the ideal-gas relation $PV = mRT$ to be

$$P_{air} = \frac{mRT}{V} = \frac{(15\text{ kg})(0.287\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(80+273)\text{K}}{1.3\text{ m}^3} = 1169\text{ kPa}$$

Then the absolute pressure in the oil pipe becomes

$$P_{oil} = P_{air} - \rho_{oil} g h_{oil} - \rho_{Hg} g h_{Hg}$$

$$= 1169\text{ kPa} - (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)\left[2.68(0.75\text{ m}) + 13.6(0.20\text{ m})\right]\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right)$$

$$= 1123\text{ kPa} \cong \mathbf{1120\ kPa}$$

(*b*) The pressure in the air tank when the temperature drops to 20°C becomes

$$P_{air} = \frac{mRT}{V} = \frac{(15\text{ kg})(0.287\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(20+273)\text{K}}{1.3\text{ m}^3} = 970\text{ kPa}$$

When the mercury level in the left arm drops a distance $x$, the rise in the mercury level in the right arm $y$ becomes

$$V_{left} = V_{right} \quad\rightarrow\quad \pi(3d)^2 x = \pi d^2 y \quad\rightarrow\quad y = 9x \quad\text{and}\quad y_{vert} = 9x\sin 50°$$

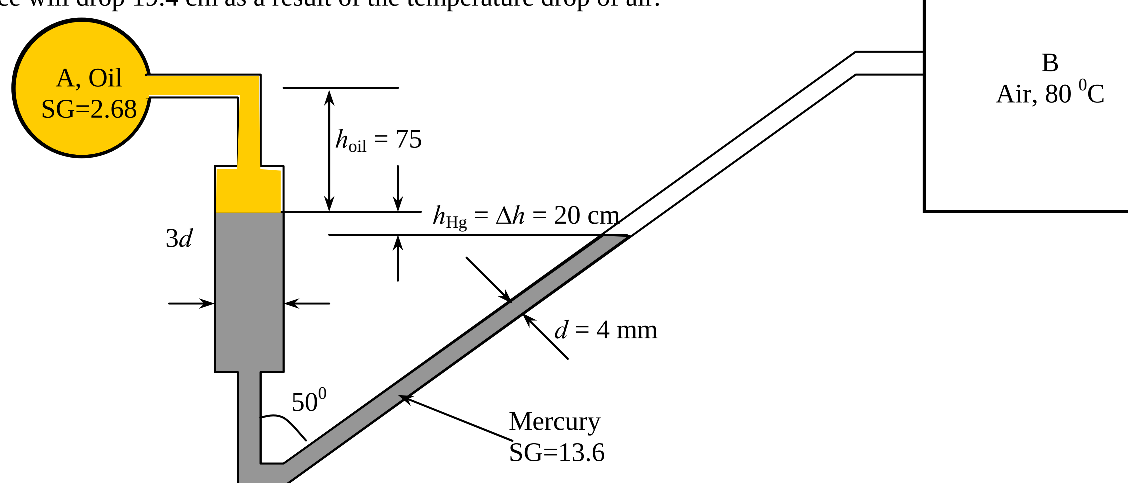
and the mercury fluid height will change by $x + 9x\sin 50°$ or 7.894$x$. Then,

$$P_{oil} + \rho_{oil} g(h_{oil} + x) + \rho_{Hg} g(h_{Hg} - 7.894x) = P_{air} \quad\rightarrow\quad \text{SG}_{oil}(h_{oil} + x) + \text{SG}_{Hg}(h_{Hg} - 7.894x) = \frac{P_{air} - P_{oil}}{\rho_w g}$$

Substituting,

$$2.68(0.75 + x) + 13.6(0.20 - 7.894x) = \frac{(970-1123)\text{ kPa}}{(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)}\left(\frac{1000\text{ kg}\cdot\text{m/s}^2}{1\text{ kN}}\right)\left(\frac{1\text{ kN/m}^2}{1\text{ kPa}}\right)$$

which yields $x = \mathbf{0.194\ m} = \mathbf{19.4\ cm}$. Therefore, the oil-mercury interface will drop 19.4 cm as a result of the temperature drop of air.




***Discussion*** Note that the pressure in constant-volume gas chambers is very sensitive to temperature changes.

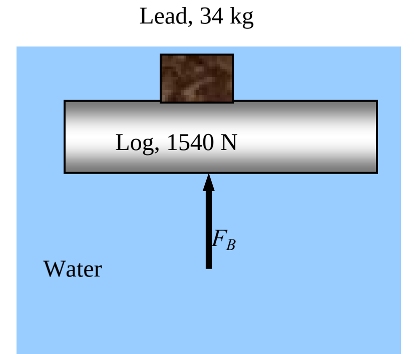
**3-136**

**Solution** The density of a wood log is to be measured by tying lead weights to it until both the log and the weights are completely submerged, and then weighing them separately in air. The average density of a given log is to be determined by this approach.

***Properties*** The density of lead weights is given to be 11,300 kg/m$^3$. We take the density of water to be 1000 kg/m$^3$.

***Analysis*** The weight of a body is equal to the buoyant force when the body is floating in a fluid while being completely submerged in it (a consequence of vertical force balance from static equilibrium). In this case the average density of the body must be equal to the density of the fluid since

$$W = F_B \quad \rightarrow \quad \rho_{\text{body}} gV = \rho_{\text{fluid}} gV \quad \rightarrow \quad \rho_{\text{body}} = \rho_{\text{fluid}}$$




Therefore,

$$\rho_{ave} = \frac{m_{\text{total}}}{V_{\text{total}}} = \frac{m_{\text{lead}} + m_{\text{log}}}{V_{\text{lead}} + V_{\text{log}}} = \rho_{\text{water}} \quad \rightarrow \quad V_{\text{log}} = V_{\text{lead}} + \frac{m_{\text{lead}} + m_{\text{log}}}{\rho_{\text{water}}}$$

where

$$V_{\text{lead}} = \frac{m_{\text{lead}}}{\rho_{\text{lead}}} = \frac{34 \text{ kg}}{11{,}300 \text{ kg/m}^3} = 3.01 \times 10^{-3} \text{ m}^3$$

$$m_{\text{log}} = \frac{W_{\text{log}}}{g} = \frac{1540 \text{ N}}{9.81 \text{ m/s}^2} \left( \frac{1 \text{ kg} \cdot \text{m/s}^2}{1 \text{ N}} \right) = 157.0 \text{ kg}$$

Substituting, the volume and density of the log are determined to be

$$V_{\text{log}} = V_{\text{lead}} + \frac{m_{\text{lead}} + m_{\text{log}}}{\rho_{\text{water}}} = 3.01 \times 10^{-3} \text{ m}^3 + \frac{(34 + 157) \text{ kg}}{1000 \text{ kg/m}^3} = \mathbf{0.194 \text{ m}^3}$$

$$\rho_{\text{log}} = \frac{m_{\text{log}}}{V_{\text{log}}} = \frac{157 \text{ kg}}{0.194 \text{ m}^3} = \mathbf{809 \text{ kg/m}^3}$$

***Discussion*** Note that the log must be completely submerged for this analysis to be valid. Ideally, the lead weights must also be completely submerged, but this is not very critical because of the small volume of the lead weights.

**3-137** **[***Also solved using EES on enclosed DVD***]**

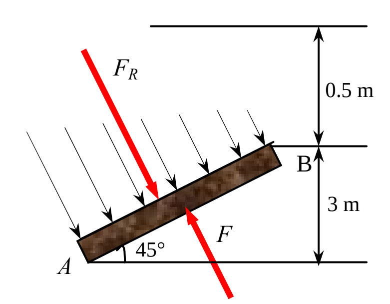
**Solution** A rectangular gate that leans against the floor with an angle of 45° with the horizontal is to be opened from its lower edge by applying a normal force at its center. The minimum force *F* required to open the water gate is to be determined.

***Assumptions*** **1** Atmospheric pressure acts on both sides of the gate, and thus it can be ignored in calculations for convenience. **2** Friction at the hinge is negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.

***Analysis*** The length of the gate and the distance of the upper edge of the gate (point *B*) from the free surface in the plane of the gate are

$$b = \frac{3\text{ m}}{\sin 45°} = 4.243\text{ m} \quad \text{and} \quad s = \frac{0.5\text{ m}}{\sin 45°} = 0.7071\text{ m}$$




The average pressure on a surface is the pressure at the centroid (midpoint) of the surface, and multiplying it by the plate area gives the resultant hydrostatic on the surface,

$$F_R = P_{avg}A = \rho g h_C A$$

$$= (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(2\text{ m})(5 \times 4.243\text{ m}^2)\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)$$

$$= 416\text{ kN}$$

The distance of the pressure center from the free surface of water along the plane of the gate is

$$y_P = s + \frac{b}{2} + \frac{b^2}{12(s+b/2)} = 0.7071 + \frac{4.243}{2} + \frac{4.243^2}{12(0.7071+4.243/2)} = 3.359\text{ m}$$

The distance of the pressure center from the hinge at point *B* is

$$L_P = y_P - s = 3.359 - 0.7071 = 2.652\text{ m}$$

Taking the moment about point *B* and setting it equal to zero gives

$$\sum M_B = 0 \quad \to \quad F_R L_P = Fb/2$$

Solving for *F* and substituting, the required force is determined to be

$$F = \frac{2F_R L_P}{b} = \frac{2(416\text{ kN})(2.652\text{ m})}{4.243\text{ m}} = \mathbf{520\text{ kN}}$$

***Discussion*** The applied force is inversely proportional to the distance of the point of application from the hinge, and the required force can be reduced by applying the force at a lower point on the gate.

**3-138**

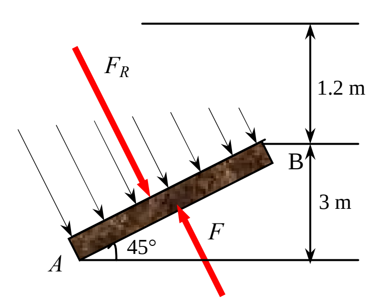
**Solution** A rectangular gate that leans against the floor with an angle of 45° with the horizontal is to be opened from its lower edge by applying a normal force at its center. The minimum force $F$ required to open the water gate is to be determined.

***Assumptions*** **1** Atmospheric pressure acts on both sides of the gate, and thus it can be ignored in calculations for convenience. **2** Friction at the hinge is negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.

***Analysis*** The length of the gate and the distance of the upper edge of the gate (point $B$) from the free surface in the plane of the gate are

$$b = \frac{3\text{ m}}{\sin 45°} = 4.243\text{ m} \quad \text{and} \quad s = \frac{1.2\text{ m}}{\sin 45°} = 1.697\text{ m}$$




The average pressure on a surface is the pressure at the centroid (midpoint) of the surface, and multiplying it by the plate area gives the resultant hydrostatic on the surface,

$$F_R = P_{avg}A = \rho g h_C A$$

$$= (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(2.7\text{ m})(5 \times 4.243\text{ m}^2)\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)$$

$$= 562\text{ kN}$$

The distance of the pressure center from the free surface of water along the plane of the gate is

$$y_P = s + \frac{b}{2} + \frac{b^2}{12(s+b/2)} = 1.697 + \frac{4.243}{2} + \frac{4.243^2}{12(1.697+4.243/2)} = 4.211\text{ m}$$

The distance of the pressure center from the hinge at point $B$ is

$$L_P = y_P - s = 4.211 - 1.697 = 2.514\text{ m}$$

Taking the moment about point $B$ and setting it equal to zero gives

$$\sum M_B = 0 \quad \rightarrow \quad F_R L_P = Fb/2$$

Solving for $F$ and substituting, the required force is determined to be

$$F = \frac{2F_R L_P}{b} = \frac{2(562\text{ N})(2.514\text{ m})}{4.243\text{ m}} = \mathbf{666\ kN}$$

***Discussion*** The applied force is inversely proportional to the distance of the point of application from the hinge, and the required force can be reduced by applying the force at a lower point on the gate.

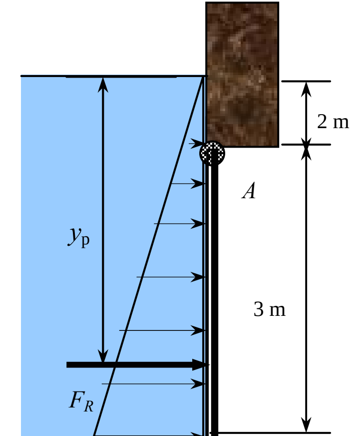
---

## 3-139

**Solution** A rectangular gate hinged about a horizontal axis along its upper edge is restrained by a fixed ridge at point *B*. The force exerted to the plate by the ridge is to be determined.

***Assumptions*** Atmospheric pressure acts on both sides of the gate, and thus it can be ignored in calculations for convenience.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.




***Analysis*** The average pressure on a surface is the pressure at the centroid (midpoint) of the surface, and multiplying it by the plate area gives the resultant hydrostatic force on the gate,

$$F_R = P_{\text{avg}} A = \rho g h_C A$$

$$= \left(1000 \text{ kg/m}^3\right)\left(9.81 \text{ m/s}^2\right)\left(3.5 \text{ m}\right)\left(3 \times 6 \text{ m}^2\right)\left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right)$$

$$= \mathbf{618 \ kN}$$

The vertical distance of the pressure center from the free surface of water is

$$y_P = s + \frac{b}{2} + \frac{b^2}{12(s+b/2)} = 2 + \frac{3}{2} + \frac{3^2}{12(2+3/2)} = \mathbf{3.71 \ m}$$

***Discussion*** You can calculate the force at point *B* required to hold back the gate by setting the net moment around hinge point *A* to zero.

## 3-140

**Solution** A rectangular gate hinged about a horizontal axis along its upper edge is restrained by a fixed ridge at point *B*. The force exerted to the plate by the ridge is to be determined.

***Assumptions*** Atmospheric pressure acts on both sides of the gate, and thus it can be ignored in calculations for convenience.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.

***Analysis*** The average pressure on a surface is the pressure at the centroid (midpoint) of the surface, and multiplying it by the wetted plate area gives the resultant hydrostatic force on the gate,

$$F_R = P_{ave} A = \rho g h_C A$$

$$= (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(1 \text{ m})[2 \times 6 \text{ m}^2]\left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right)$$

$$= \mathbf{118 \ kN}$$

The vertical distance of the pressure center from the free surface of water is

$$y_P = \frac{2h}{3} = \frac{2(2 \text{ m})}{3} = \mathbf{1.33 \ m}$$

***Discussion*** Compared to the previous problem (with higher water depth), the force is much smaller, as expected. Also, the center of pressure on the gate is much lower (closer to the ground) for the case with the lower water depth.

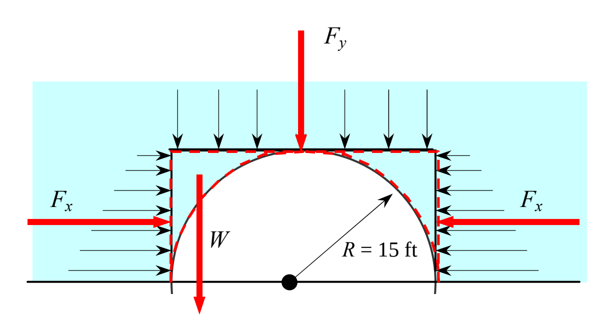
**3-141E**

**Solution** A semicircular tunnel is to be built under a lake. The total hydrostatic force acting on the roof of the tunnel is to be determined.

***Assumptions*** Atmospheric pressure acts on both sides of the tunnel, and thus it can be ignored in calculations for convenience.

***Properties*** We take the density of water to be 62.4 lbm/ft$^3$ throughout.

***Analysis*** We consider the free body diagram of the liquid block enclosed by the circular surface of the tunnel and its vertical (on both sides) and horizontal projections. The hydrostatic forces acting on the vertical and horizontal plane surfaces as well as the weight of the liquid block are determined as follows:




Horizontal force on vertical surface (each side):

$$F_H = F_x = P_{avg}A = \rho g h_C A = \rho g (s + R/2) A$$

$$= (62.4 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)(135 + 15/2 \text{ ft})(15 \text{ ft} \times 800 \text{ ft})\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm}\cdot\text{ft/s}^2}\right)$$

$$= 1.067 \times 10^8 \text{ lbf (on each side of the tunnel)}$$

Vertical force on horizontal surface (*downward*):

$$F_y = P_{avg}A = \rho g h_C A = \rho g h_{top} A$$

$$= (62.4 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)(135 \text{ ft})(30 \text{ ft} \times 800 \text{ ft})\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm}\cdot\text{ft/s}^2}\right)$$

$$= 2.022 \times 10^8 \text{ lbf}$$

Weight of fluid block on each side within the control volume *(downward):*

$$W = mg = \rho g V = \rho g (R^2 - \pi R^2/4)(2000 \text{ ft})$$

$$= (62.4 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)(15 \text{ ft})^2(1 - \pi/4)(800 \text{ ft})\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm}\cdot\text{ft/s}^2}\right)$$

$$= 2.410 \times 10^6 \text{ lbf (on each side)}$$

Therefore, the net downward vertical force is

$$F_V = F_y + 2W = 2.022 \times 10^8 + 2 \times 0.02410 \times 10^6 = \mathbf{2.07 \times 10^8 \text{ lbf}}$$

This is also the **net force** acting on the tunnel since the horizontal forces acting on the right and left side of the tunnel cancel each other since they are equal and opposite.

***Discussion*** The weight of the two water bocks on the sides represents only about 2.4% of the total vertical force on the tunnel. Therefore, to obtain a reasonable first approximation for deep tunnels, these volumes can be neglected, yielding $F_V$ = 2.02 × 10$^8$ lbf. A more conservative approximation would be to estimate the force on the *bottom* of the lake if the tunnel were not there. This yields $F_V$ = 2.25 × 10$^8$ lbf. The actual force is between these two estimates, as expected.

**3-142**

**Solution** A hemispherical dome on a level surface filled with water is to be lifted by attaching a long tube to the top and filling it with water. The required height of water in the tube to lift the dome is to be determined.

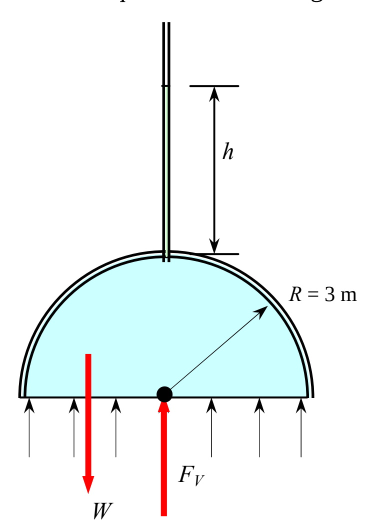
***Assumptions*** **1** Atmospheric pressure acts on both sides of the dome, and thus it can be ignored in calculations for convenience. **2** The weight of the tube and the water in it is negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.

***Analysis*** We take the dome and the water in it as the system. When the dome is about to rise, the reaction force between the dome and the ground becomes zero. Then the free body diagram of this system involves the weights of the dome and the water, balanced by the hydrostatic pressure force from below. Setting these forces equal to each other gives

$$\sum F_y = 0: \quad F_V = W_{dome} + W_{water}$$

$$\rho g(h+R)\pi R^2 = m_{dome} g + m_{water} g$$




Solving for $h$ gives

$$h = \frac{m_{dome} + m_{water}}{\rho \pi R^2} - R = \frac{m_{dome} + \rho[4\pi R^3/6]}{\rho \pi R^2} - R$$

Substituting,

$$h = \frac{(50{,}000\ \text{kg}) + 4\pi(1000\ \text{kg/m}^3)(3\ \text{m})^3/6}{(1000\ \text{kg/m}^3)\pi(3\ \text{m})^2} - (3\ \text{m}) = \mathbf{0.77\ m}$$

Therefore, this dome can be lifted by attaching a tube which is 77 cm long.

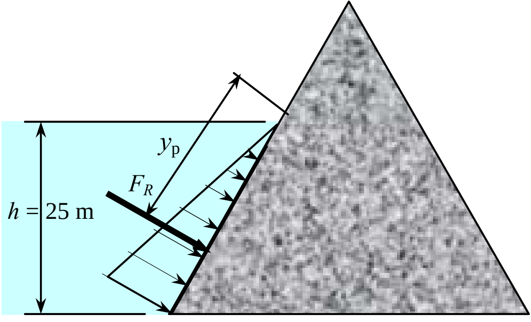
***Discussion*** Note that the water pressure in the dome can be changed greatly by a small amount of water in the vertical tube.

**3-143**

**Solution** The water in a reservoir is restrained by a triangular wall. The total force (hydrostatic + atmospheric) acting on the inner surface of the wall and the horizontal component of this force are to be determined.

***Assumptions*** **1** Atmospheric pressure acts on both sides of the gate, and thus it can be ignored in calculations for convenience. **2** Friction at the hinge is negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$ throughout.




***Analysis*** The length of the wall surface underwater is

$$b = \frac{25\text{ m}}{\sin 60°} = 28.87\text{ m}$$

The average pressure on a surface is the pressure at the centroid (midpoint) of the surface, and multiplying it by the plate area gives the resultant hydrostatic force on the surface,

$$F_R = P_{avg}A = (P_{atm} + \rho g h_C)A$$
$$= \left[100{,}000\text{ N/m}^2 + (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(12.5\text{ m})\right](150 \times 28.87\text{ m}^2)\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)$$
$$= \mathbf{9.64 \times 10^8\ N}$$

Noting that

$$\frac{P_0}{\rho g \sin 60°} = \frac{100{,}000\text{ N/m}^2}{(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)\sin 60°}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right) = 11.77\text{ m}$$

the distance of the pressure center from the free surface of water along the wall surface is

$$y_p = s + \frac{b}{2} + \frac{b^2}{12\left(s + \dfrac{b}{2} + \dfrac{P_0}{\rho g \sin\theta}\right)} = 0 + \frac{28.87\text{ m}}{2} + \frac{(28.87\text{ m})^2}{12\left(0 + \dfrac{28.87\text{ m}}{2} + 11.77\text{ m}\right)} = \mathbf{17.1\,m}$$

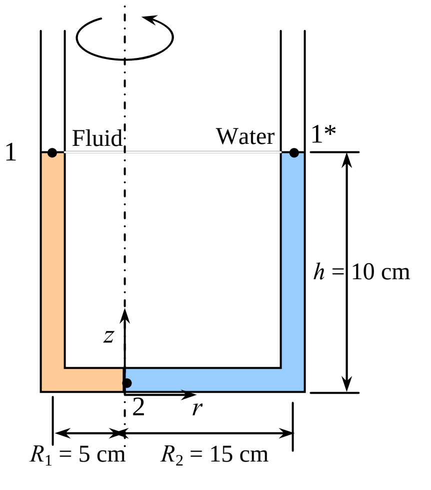
The magnitude of the horizontal component of the hydrostatic force is simply $F_R \sin\theta$,

$$F_H = F_R \sin\theta = (9.64 \times 10^8\text{ N})\sin 60° = \mathbf{8.35 \times 10^8\ N}$$

***Discussion*** Atmospheric pressure is usually ignored in the analysis for convenience since it acts on both sides of the walls.

**3-144**

**Solution** A U-tube that contains water in its right arm and another liquid in its left arm is rotated about an axis closer to the left arm. For a known rotation rate at which the liquid levels in both arms are the same, the density of the fluid in the left arm is to be determined.




***Assumptions*** **1** Both the fluid and the water are incompressible fluids. **2** The two fluids meet at the axis of rotation, and thus there is only water to the right of the axis of rotation.

***Properties*** We take the density of water to be 1000 kg/m³.

***Analysis*** The pressure difference between two points 1 and 2 in an incompressible fluid rotating in rigid body motion (the *same* fluid) is given by

$$P_2 - P_1 = \frac{\rho\omega^2}{2}(r_2^2 - r_1^2) - \rho g(z_2 - z_1)$$

where

$$\omega = 2\pi\dot{n} = 2\pi(30\text{ rev/min})\left(\frac{1\text{ min}}{60\text{ s}}\right) = 3.14\text{ rad/s}$$

(for both arms of the U-tube).

The pressure at point 2 is the same for both fluids, so are the pressures at points 1 and 1* ($P_1 = P_1* = P_{atm}$). Therefore, $P_2 - P_1$ is the same for both fluids. Noting that $z_2 - z_1 = -h$ for both fluids and expressing $P_2 - P_1$ for each fluid,

$$Water: \quad P_2 - P_1* = \frac{\rho_w\omega^2}{2}(0 - R_2^2) - \rho_w g(-h) = \rho_w(-\omega^2 R_2^2/2 + gh)$$

$$Fluid: \quad P_2 - P_1 = \frac{\rho_f\omega^2}{2}(0 - R_1^2) - \rho_f g(-h) = \rho_f(-\omega^2 R_1^2/2 + gh)$$

Setting them equal to each other and solving for $\rho_f$ gives

$$\rho_f = \frac{-\omega^2 R_2^2/2 + gh}{-\omega^2 R_1^2/2 + gh}\rho_w = \frac{-(3.14\text{ rad/s})^2(0.15\text{ m})^2 + (9.81\text{ m/s}^2)(0.10\text{ m})}{-(3.14\text{ rad/s})^2(0.05\text{ m})^2 + (9.81\text{ m/s}^2)(0.10\text{ m})}(1000\text{ kg/m}^3) = \mathbf{794\ kg/m^3}$$

***Discussion*** Note that this device can be used to determine relative densities, though it wouldn't be very practical.

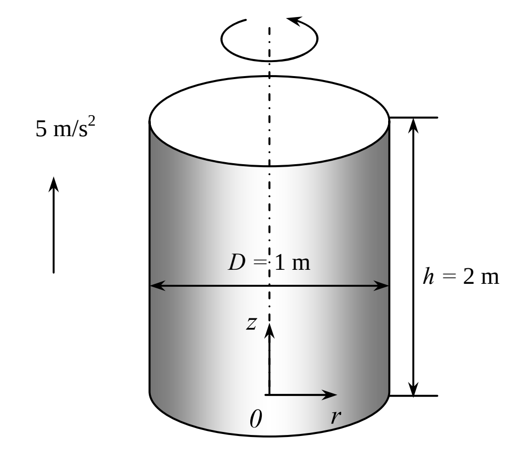
---

**3-145**

**Solution** A vertical cylindrical tank is completely filled with gasoline, and the tank is rotated about its vertical axis at a specified rate while being accelerated upward. The pressures difference between the centers of the bottom and top surfaces, and the pressures difference between the center and the edge of the bottom surface are to be determined.

***Assumptions*** **1** The increase in the rotational speed is very slow so that the liquid in the container always acts as a rigid body. **2** Gasoline is an incompressible substance.

***Properties*** The density of the gasoline is given to be 740 kg/m$^3$.




***Analysis*** The pressure difference between two points 1 and 2 in an incompressible fluid rotating in rigid body motion is given by $P_2 - P_1 = \frac{\rho\omega^2}{2}(r_2^2 - r_1^2) - \rho g(z_2 - z_1)$. The effect of linear acceleration in the vertical direction is accounted for by replacing $g$ by $g + a_z$. Then,

$$P_2 - P_1 = \frac{\rho\omega^2}{2}(r_2^2 - r_1^2) - \rho(g + a_z)(z_2 - z_1)$$

where $R$ = 0.50 m is the radius, and

$$\omega = 2\pi\dot{n} = 2\pi(90\ \text{rev/min})\left(\frac{1\ \text{min}}{60\ \text{s}}\right) = 9.425\ \text{rad/s}$$

($a$) Taking points 1 and 2 to be the centers of the bottom and top surfaces, respectively, we have $r_1 = r_2 = 0$ and $z_2 - z_1 = h = 3\ \text{m}$. Then,

$$P_{\text{center, top}} - P_{\text{center, bottom}} = 0 - \rho(g + a_z)(z_2 - z_1) = -\rho(g + a_z)h$$

$$= -(740\ \text{kg/m}^3)(9.81\ \text{m/s}^2 + 5)(2\ \text{m})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) = 21.8\ \text{kN/m}^2 = \mathbf{21.9\ \ kPa}$$

($b$) Taking points 1 and 2 to be the center and edge of the bottom surface, respectively, we have $r_1 = 0$, $r_2 = R$, and $z_2 = z_1 = 0$. Then,

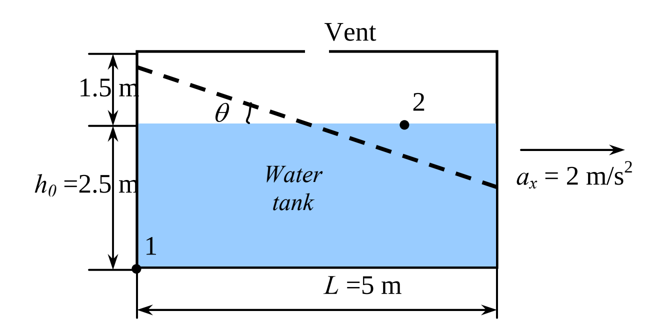
$$P_{\text{edge, bottom}} - P_{\text{center, bottom}} = \frac{\rho\omega^2}{2}(R_2^2 - 0) - 0 = \frac{\rho\omega^2 R^2}{2}$$

$$= \frac{(740\ \text{kg/m}^3)(9.425\ \text{rad/s})^2(0.50\ \text{m})^2}{2}\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) = 8.22\text{kN/m}^2 = \mathbf{8.22\ \ kPa}$$

***Discussion*** Note that the rotation of the tank does not affect the pressure difference along the axis of the tank. Likewise, the vertical acceleration does not affect the pressure difference between the edge and the center of the bottom surface (or any other horizontal plane).

**3-146**

**Solution** A rectangular water tank open to the atmosphere is accelerated to the right on a level surface at a specified rate. The maximum pressure in the tank above the atmospheric level is to be determined.




***Assumptions*** **1** The road is horizontal during acceleration so that acceleration has no vertical component ($a_z$ = 0). **2** Effects of splashing, breaking and driving over bumps are assumed to be secondary, and are not considered. **3** The vent is never blocked, and thus the minimum pressure is the atmospheric pressure.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** We take the *x*-axis to be the direction of motion, the *z*-axis to be the upward vertical direction. The tangent of the angle the free surface makes with the horizontal is

$$\tan\theta = \frac{a_x}{g + a_z} = \frac{2}{9.81 + 0} = 0.2039 \quad \text{(and thus } \theta = 11.5°\text{)}$$

The maximum vertical rise of the free surface occurs at the back of the tank, and the vertical midsection experiences no rise or drop during acceleration. Then the maximum vertical rise at the back of the tank relative to the neutral midplane is

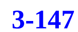
$$\Delta z_{\max} = (L/2)\tan\theta = [(5\text{ m})/2] \times 0.2039 = 0.510\text{ m}$$

which is less than 1.5 m high air space. Therefore, water never reaches the ceiling, and the maximum water height and the corresponding maximum pressure are

$$h_{\max} = h_0 + \Delta z_{\max} = 2.50 + 0.510 = 3.01\text{ m}$$

$$P_{\max} = P_1 = \rho g h_{\max} = (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(3.01\text{ m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 29.5\text{ kN/m}^2 = \mathbf{29.5\ kPa}$$

***Discussion*** It can be shown that the gage pressure at the bottom of the tank varies from 29.5 kPa at the back of the tank to 24.5 kPa at the midsection and 19.5 kPa at the front of the tank.



**3-147** 

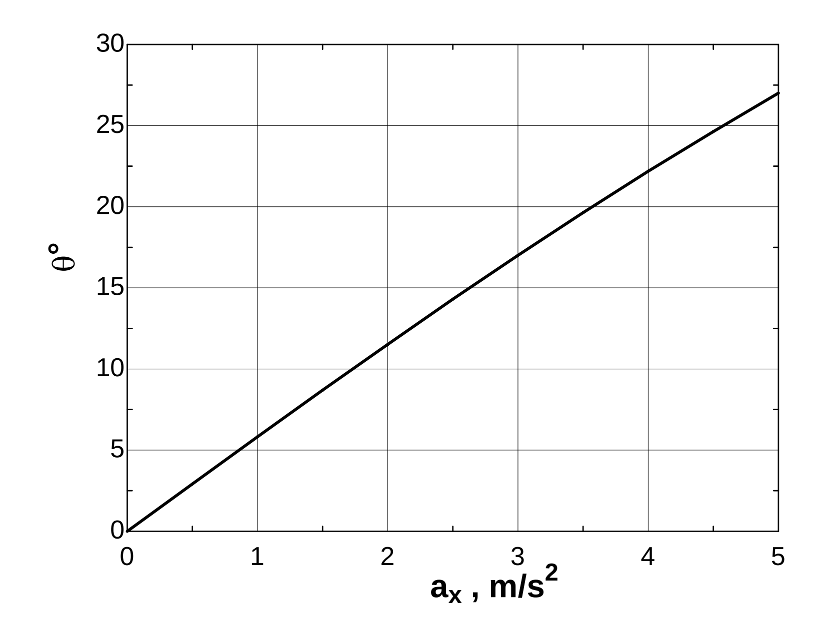
**Solution** The previous problem is reconsidered. The effect of acceleration on the slope of the free surface of water in the tank as the acceleration varies from 0 to 5 m/s$^2$ in increments of 0.5 m/s$^2$ is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
g=9.81 "m/s2"
rho=1000 "kg/m3"
L=5 "m"
h0=2.5 "m"

a_z=0
tan(theta)=a_x/(g+a_z)
h_max=h0+(L/2)*tan(theta)
P_max=rho*g*h_max/1000 "kPa"
```

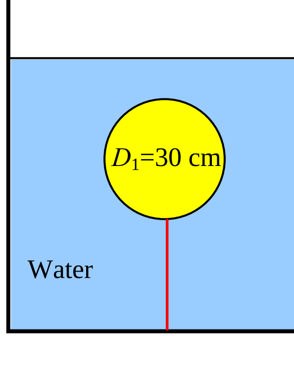
| Acceleration $a_x$, m/s$^2$ | Free surface angle, $\theta$° | Maximum height $h_{max}$, m | Maximum pressure $P_{max}$, kPa |
|---|---|---|---|
| 0.0 | 0.0 | 2.50 | 24.5 |
| 0.5 | 2.9 | 2.63 | 25.8 |
| 1.0 | 5.8 | 2.75 | 27.0 |
| 1.5 | 8.7 | 2.88 | 28.3 |
| 2.0 | 11.5 | 3.01 | 29.5 |
| 2.5 | 14.3 | 3.14 | 30.8 |
| 3.0 | 17.0 | 3.26 | 32.0 |
| 3.5 | 19.6 | 3.39 | 33.3 |
| 4.0 | 22.2 | 3.52 | 34.5 |
| 4.5 | 24.6 | 3.65 | 35.8 |
| 5.0 | 27.0 | 3.77 | 37.0 |




***Discussion*** Note that water never reaches the ceiling, and a full free surface is formed in the tank.

**3-148**

**Solution** An elastic air balloon submerged in water is attached to the base of the tank. The change in the tension force of the cable is to be determined when the tank pressure is increased and the balloon diameter is decreased in accordance with the relation $P = CD^{-2}$.




***Assumptions*** **1** Atmospheric pressure acts on all surfaces, and thus it can be ignored in calculations for convenience. **2** Water is an incompressible fluid. **3** The weight of the balloon and the air in it is negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** The tension force on the cable holding the balloon is determined from a force balance on the balloon to be

$$F_{cable} = F_B - W_{balloon} \cong F_B$$

The buoyancy force acting on the balloon initially is

$$F_{B,1} = \rho_w g V_{balloon,1} = \rho_w g \frac{\pi D_1^3}{6} = (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)\frac{\pi(0.30 \text{ m})^3}{6}\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = 138.7 \text{ N}$$

The variation of pressure with diameter is given as $P = CD^{-2}$, which is equivalent to $D = \sqrt{C/P}$. Then the final diameter of the ball becomes

$$\frac{D_2}{D_1} = \frac{\sqrt{C/P_2}}{\sqrt{C/P_1}} = \sqrt{\frac{P_1}{P_2}} \quad \rightarrow \quad D_2 = D_1\sqrt{\frac{P_1}{P_2}} = (0.30 \text{ m})\sqrt{\frac{0.1 \text{ MPa}}{1.6 \text{ MPa}}} = 0.075 \text{ m}$$

The buoyancy force acting on the balloon in this case is

$$F_{B,2} = \rho_w g V_{balloon,2} = \rho_w g \frac{\pi D_2^3}{6} = (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)\frac{\pi(0.075 \text{ m})^3}{6}\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = 2.2 \text{ N}$$

Then the percent change in the cable for becomes

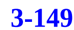
$$Change\% = \frac{F_{cable,1} - F_{cable,2}}{F_{cable,1}} * 100 = \frac{F_{B,1} - F_{B,2}}{F_{B,1}} * 100 = \frac{138.7 - 2.2}{138.7} * 100 = \mathbf{98.4\%}.$$

Therefore, increasing the tank pressure in this case results in 98.4% reduction in cable tension.

***Discussion*** We can obtain a relation for the change in cable tension as follows:

$$Change\% = \frac{F_{B,1} - F_{B,2}}{F_{B,1}} * 100 = \frac{\rho_w g V_{\text{balloon},1} - \rho_w g V_{\text{balloon},2}}{\rho_w g V_{\text{balloon},1}} * 100$$

$$= 100\left(1 - \frac{V_{\text{balloon},2}}{V_{\text{balloon},1}}\right) = 100\left(1 - \frac{D_2^3}{D_1^3}\right) = 100\left(1 - \left(\frac{P_1}{P_2}\right)^{3/2}\right)$$



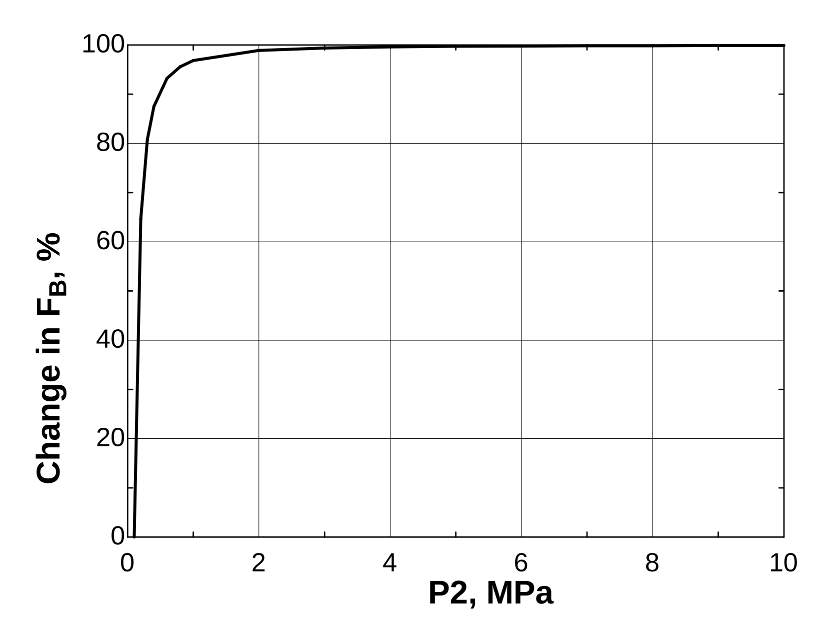
**3-149** 

**Solution** The previous problem is reconsidered. The effect of the air pressure above the water on the cable force as the pressure varies from 0.1 MPa to 10 MPa is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
P1=0.1 "MPa"
Change=100*(1-(P1/P2)^1.5)
```

| Tank pressure $P_2$, MPa | %Change in cable tension |
|---|---|
| 0.1 | 0.0 |
| 0.2 | 64.6 |
| 0.3 | 80.8 |
| 0.4 | 87.5 |
| 0.6 | 93.2 |
| 0.8 | 95.6 |
| 1 | 96.8 |
| 2 | 98.9 |
| 3 | 99.4 |
| 4 | 99.6 |
| 5 | 99.7 |
| 6 | 99.8 |
| 7 | 99.8 |
| 8 | 99.9 |
| 9 | 99.9 |
| 10 | 99.9 |




***Discussion*** The change in cable tension is at first very rapid, but levels off as the balloon shrinks to nearly zero diameter at high pressure.

---

**3-150**

**Solution** An iceberg floating in seawater is considered. The volume fraction of the iceberg submerged in seawater is to be determined, and the reason for their turnover is to be explained.

***Assumptions*** **1** The buoyancy force in air is negligible. **2** The density of iceberg and seawater are uniform.

***Properties*** The densities of iceberg and seawater are given to be 917 kg/m$^3$ and 1042 kg/m$^3$, respectively.

***Analysis*** (*a*) The weight of a body floating in a fluid is equal to the buoyant force acting on it (a consequence of vertical force balance from static equilibrium). Therefore,

$$W = F_B$$

$$\rho_{\text{body}} g V_{\text{total}} = \rho_{\text{fluid}} g V_{\text{submerged}}$$

$$\frac{V_{\text{submerged}}}{V_{\text{total}}} = \frac{\rho_{\text{body}}}{\rho_{\text{fluid}}} = \frac{\rho_{\text{iceberg}}}{\rho_{\text{seawater}}} = \frac{917}{1042} = 0.880 \quad \text{or} \quad \mathbf{88\%}$$

Therefore, 88% of the volume of the iceberg is submerged in this case.

(*b*) Heat transfer to the iceberg due to the temperature difference between the seawater and an iceberg causes uneven melting of the irregularly shaped iceberg. The resulting **shift in the center of mass causes the iceberg to turn over**.

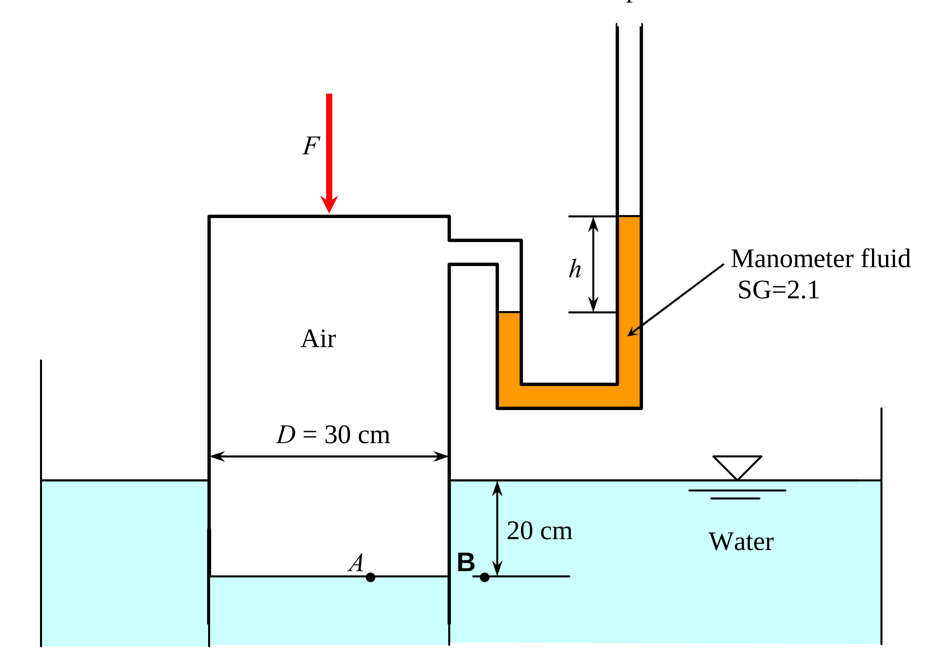
Sea
Iceberg
*W*
$F_B$

***Discussion*** The submerged fraction depends on the density of seawater, and this fraction can differ in different seas.

---

**3-151**

**Solution** A cylindrical container equipped with a manometer is inverted and pressed into water. The differential height of the manometer and the force needed to hold the container in place are to be determined.




***Assumptions*** **1** Atmospheric pressure acts on all surfaces, and thus it can be ignored in calculations for convenience. **2** The variation of air pressure inside cylinder is negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$. The density of the manometer fluid is

$$\rho_{\text{mano}} = \text{SG} \times \rho_w = 2.1\left(1000 \text{ kg/m}^3\right) = 2100 \text{ kg/m}^3$$

***Analysis*** The pressures at point $A$ and $B$ must be the same since they are on the same horizontal line in the same fluid. Then the gage pressure in the cylinder becomes

$$P_{\text{air, gage}} = \rho_{\text{w}} g h_{\text{w}} \quad = (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(0.20 \text{ m})\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = 1962 \text{ N/m}^2 = 1962 \text{ Pa}$$

The manometer also indicates the gage pressure in the cylinder. Therefore,

$$P_{\text{air, gage}} = \left(\rho g h\right)_{\text{mano}} \quad \rightarrow \quad h = \frac{P_{\text{air, gage}}}{\rho_{\text{mano}} g} = \frac{1962 \text{ N/m}^2}{(2100 \text{ kg/m}^3)(9.81 \text{ m/s}^2)}\left(\frac{1 \text{ kg}\cdot\text{m/s}^2}{1 \text{ kN/m}^2}\right) = 0.0950 \text{ m} = \mathbf{9.50\ cm}$$

A force balance on the cylinder in the vertical direction yields

$$F + W = P_{\text{air, gage}} A_c$$

Solving for $F$ and substituting,

$$F = P_{\text{air, gage}} \frac{\pi D^2}{4} - W = \left(1962 \text{ N/m}^2\right)\frac{\pi\left(0.30 \text{ m}\right)^2}{4} - 79 \text{ N} = \mathbf{59.7\ N}$$

***Discussion*** We could also solve this problem by considering the atmospheric pressure, but we would obtain the same result since atmospheric pressure would cancel out.

## Design and Essay Problems

**3-152**

**Solution** We are to discuss the design of shoes that enable people to walk on water.

***Discussion*** Students' discussions should be unique and will differ from each other.

**3-153**

**Solution** We are to discuss how to measure the volume of a rock without using any volume measurement devices.

***Analysis*** The volume of a rock can be determined without using any volume measurement devices as follows: We weigh the rock in the air and then in the water. The difference between the two weights is due to the buoyancy force, which is equal to $F_B = \rho_{water} g V_{body}$. Solving this relation for $V_{body}$ gives the volume of the rock.

***Discussion*** Since this is an open-ended design problem, students may come up with different, but equally valid techniques.



# Solutions Manual for

# Fluid Mechanics: Fundamentals and Applications

# by Çengel & Cimbala

---

# CHAPTER 4
# FLUID KINEMATICS

---

## Introductory Problems

### 4-1C

**Solution** We are to define and explain kinematics and fluid kinematics.

***Analysis*** *Kinematics* means **the study of *motion***. *Fluid kinematics* is the **study of how fluids flow and how to describe fluid motion**. Fluid kinematics deals with describing the motion of fluids without considering (or even understanding) the forces and moments that *cause* the motion.

***Discussion*** Fluid kinematics deals with such things as describing how a fluid particle translates, distorts, and rotates, and how to visualize flow fields.

---

### 4-2

**Solution** We are to write an equation for centerline speed through a nozzle, given that the flow speed increases parabolically.

***Assumptions*** **1** The flow is steady. **2** The flow is axisymmetric. **3** The water is incompressible.

***Analysis*** A general equation for a parabola in the $x$ direction is

*General parabolic equation*: $$u = a + b\left(x - c\right)^2 \qquad (1)$$

We have two boundary conditions, namely at $x = 0$, $u = u_{\text{entrance}}$ and at $x = L$, $u = u_{\text{exit}}$. By inspection, Eq. 1 is satisfied by setting $c = 0$, $a = u_{\text{entrance}}$ and $b = (u_{\text{exit}} - u_{\text{entrance}})/L^2$. Thus, Eq. 1 becomes

*Parabolic speed*: $$\boxed{u = u_{\text{entrance}} + \frac{\left(u_{\text{exit}} - u_{\text{entrance}}\right)}{L^2}x^2} \qquad (2)$$

***Discussion*** You can verify Eq. 2 by plugging in $x = 0$ and $x = L$.

---

### 4-3

**Solution** For a given velocity field we are to find out if there is a stagnation point. If so, we are to calculate its location.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $x$-$y$ plane.

***Analysis*** The velocity field is

$$\vec{V} = \left(u, v\right) = \left(0.5 + 1.2x\right)\vec{i} + \left(-2.0 - 1.2y\right)\vec{j} \qquad (1)$$

At a stagnation point, both $u$ and $v$ must equal zero. At any point ($x$,$y$) in the flow field, the velocity components $u$ and $v$ are obtained from Eq. 1,

*Velocity components*: $$u = 0.5 + 1.2x \qquad v = -2.0 - 1.2y \qquad (2)$$

Setting these to zero yields

*Stagnation point*: $$\begin{aligned} 0 &= 0.5 + 1.2x & x &= -0.4167 \\ 0 &= -2.0 - 1.2y & y &= -1.667 \end{aligned} \qquad (3)$$

So, **yes there is a stagnation point**; its location is $\boldsymbol{x = -0.417, y = -1.67}$ (to 3 digits).

***Discussion*** If the flow were three-dimensional, we would have to set $w = 0$ as well to determine the location of the stagnation point. In some flow fields there is more than one stagnation point.

---

**4-4**

**Solution** For a given velocity field we are to find out if there is a stagnation point. If so, we are to calculate its location.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity field is

$$\vec{V} = (u, v) = \left(a^2 - (b - cx)^2\right)\vec{i} + \left(-2cby + 2c^2xy\right)\vec{j} \qquad (1)$$

At a stagnation point, both *u* and *v* must equal zero. At any point (*x*,*y*) in the flow field, the velocity components *u* and *v* are obtained from Eq. 1,

*Velocity components*: $$u = a^2 - (b - cx)^2 \qquad v = -2cby + 2c^2xy \qquad (2)$$

Setting these to zero and solving simultaneously yields

*Stagnation point*: $$\begin{aligned} 0 &= a^2 - (b - cx)^2 \qquad & x &= \frac{b-a}{c} \\ v &= -2cby + 2c^2xy \qquad & y &= 0 \end{aligned} \qquad (3)$$

So, **yes there is a stagnation point**; its location is ***x* = (*b* – *a*)/*c*, *y* = 0**.

***Discussion*** If the flow were three-dimensional, we would have to set *w* = 0 as well to determine the location of the stagnation point. In some flow fields there is more than one stagnation point.

---

## Lagrangian and Eulerian Descriptions

**4-5C**

**Solution** We are to define the Lagrangian description of fluid motion.

***Analysis*** In the *Lagrangian description* of fluid motion, **individual fluid particles** (fluid elements composed of a fixed, identifiable mass of fluid) **are followed**.

***Discussion*** The Lagrangian method of studying fluid motion is similar to that of studying billiard balls and other solid objects in physics.

---

**4-6C**

**Solution** We are to compare the Lagrangian method to the study of systems and control volumes and determine to which of these it is most similar.

***Analysis*** **The Lagrangian method is more similar to system analysis** (i.e., *closed* system analysis). In both cases, we follow a mass of fixed identity as it moves in a flow. In a control volume analysis, on the other hand, mass moves into and out of the control volume, and we don't follow any particular chunk of fluid. Instead we analyze whatever fluid happens to be inside the control volume at the time.

***Discussion*** In fact, the Lagrangian analysis is the same as a system analysis in the limit as the size of the system shrinks to a point.

---

**4-7C**

**Solution** We are to define the Eulerian description of fluid motion, and explain how it differs from the Lagrangian description.

***Analysis*** In the *Eulerian description* of fluid motion, we are concerned with ***field variables*, such as velocity, pressure, temperature, etc., as functions of space and time within a flow domain or control volume**. In contrast to the Lagrangian method, fluid flows into and out of the Eulerian flow domain, and we do not keep track of the motion of particular identifiable fluid particles.

***Discussion*** The Eulerian method of studying fluid motion is not as "natural" as the Lagrangian method since the fundamental conservation laws apply to moving particles, not to fields.

---

**4-8C**

**Solution** We are to determine whether a measurement is Lagrangian or Eulerian.

***Analysis*** Since the probe is fixed in space and the fluid flows around it, we are *not* following individual fluid particles as they move. Instead, we are measuring a field variable at a particular location in space. Thus this is an **Eulerian measurement**.

***Discussion*** If a neutrally buoyant probe were to move with the flow, its results would be Lagrangian measurements – following fluid particles.

---

**4-9C**

**Solution** We are to determine whether a measurement is Lagrangian or Eulerian.

***Analysis*** Since the probe moves with the flow and is neutrally buoyant, we are following individual fluid particles as they move through the pump. Thus this is a **Lagrangian measurement**.

***Discussion*** If the probe were instead fixed at one location in the flow, its results would be Eulerian measurements.

---

**4-10C**

**Solution** We are to determine whether a measurement is Lagrangian or Eulerian.

***Analysis*** Since the weather balloon moves with the air and is neutrally buoyant, we are following individual "fluid particles" as they move through the atmosphere. Thus this is a **Lagrangian measurement**. Note that in this case the "fluid particle" is huge, and can follow gross features of the flow – the balloon obviously cannot follow small scale turbulent fluctuations in the atmosphere.

***Discussion*** When weather monitoring instruments are mounted on the roof of a building, the results are Eulerian measurements.

---

**4-11C**

**Solution** We are to determine whether a measurement is Lagrangian or Eulerian.

***Analysis*** Relative to the airplane, the probe is fixed and the air flows around it. We are *not* following individual fluid particles as they move. Instead, we are measuring a field variable at a particular location in space relative to the moving airplane. Thus this is an **Eulerian measurement**.

***Discussion*** The airplane is moving, but it is not moving with the flow.

---

### 4-12C

**Solution** We are to compare the Eulerian method to the study of systems and control volumes and determine to which of these it is most similar.

***Analysis*** **The Eulerian method is more similar to control volume analysis**. In both cases, mass moves into and out of the flow domain or control volume, and we don't follow any particular chunk of fluid. Instead we analyze whatever fluid happens to be inside the control volume at the time.

***Discussion*** In fact, the Eulerian analysis is the same as a control volume analysis except that Eulerian analysis is usually applied to infinitesimal volumes and differential equations of fluid flow, whereas control volume analysis usually refers to finite volumes and integral equations of fluid flow.

---

### 4-13C

**Solution** We are to define a steady flow field in the Eulerian description, and discuss particle acceleration in such a flow.

***Analysis*** A flow field is defined as *steady* in the Eulerian frame of reference when **properties at any point in the flow field do not change with respect to time**. In such a flow field, individual fluid particles may still experience non-zero acceleration – the answer to the question is **yes**.

***Discussion*** Although velocity is not a function of time in a steady flow field, its total derivative with respect to time $\left(\vec{a} = d\vec{V} / dt\right)$ is not necessarily zero since the acceleration is composed of a local (unsteady) part which is zero and an advective part which is not necessarily zero.

---

### 4-14C

**Solution** We are to list three alternate names for material derivative.

***Analysis*** The material derivative is also called **total derivative**, **particle derivative**, **Eulerian derivative**, **Lagrangian derivative,** and **substantial derivative**. "Total" is appropriate because the material derivative includes both local (unsteady) and convective parts. "Particle" is appropriate because it stresses that the material derivative is one following fluid particles as they move about in the flow field. "Eulerian" is appropriate since the material derivative is used to transform from Lagrangian to Eulerian reference frames. "Lagrangian" is appropriate since the material derivative is used to transform from Lagrangian to Eulerian reference frames. Finally, "substantial" is not as clear of a term for the material derivative, and we are not sure of its origin.

***Discussion*** All of these names emphasize that we are following a fluid particle as it moves through a flow field.

---

## 4-15

**Solution** We are to calculate the material acceleration for a given velocity field.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity field is

$$\vec{V} = (u, v) = (U_0 + bx)\vec{i} - by\vec{j} \qquad (1)$$

The acceleration field components are obtained from its definition (the material acceleration) in Cartesian coordinates,

$$\begin{aligned} a_x &= \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = 0 + (U_0 + bx)b + (-by)0 + 0 \\ a_y &= \frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = 0 + (U_0 + bx)0 + (-by)(-b) + 0 \end{aligned} \qquad (2)$$

where the unsteady terms are zero since this is a steady flow, and the terms with *w* are zero since the flow is two-dimensional. Eq. 2 simplifies to

*Material acceleration components*: $$\boxed{a_x = b(U_0 + bx) \qquad a_y = b^2 y} \qquad (3)$$

In terms of a vector,

*Material acceleration vector*: $$\boxed{\vec{a} = b(U_0 + bx)\vec{i} + b^2 y\vec{j}} \qquad (4)$$

***Discussion*** For positive *x* and *b*, fluid particles accelerate in the positive *x* direction. Even though this flow is steady, there is still a non-zero acceleration field.

---

## 4-16

**Solution** For a given pressure and velocity field, we are to calculate the rate of change of pressure following a fluid particle.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The pressure field is

*Pressure field*: $$P = P_0 - \frac{\rho}{2}\left[2U_0 bx + b^2\left(x^2 + y^2\right)\right] \qquad (1)$$

By definition, the material derivative, when applied to pressure, produces the rate of change of pressure following a fluid particle. Using Eq. 1 and the velocity components from the previous problem,

$$\begin{aligned} \frac{DP}{Dt} &= \underbrace{\cancel{\frac{\partial P}{\partial t}}}_{\text{Steady}} + u\frac{\partial P}{\partial x} + v\frac{\partial P}{\partial y} + \underbrace{\cancel{w\frac{\partial P}{\partial z}}}_{\text{Two-dimensional}} \\ &= (U_0 + bx)\left(-\rho U_0 b - \rho b^2 x\right) + (-by)\left(-\rho b^2 y\right) \end{aligned} \qquad (2)$$

where the unsteady term is zero since this is a steady flow, and the term with *w* is zero since the flow is two-dimensional. Eq. 2 simplifies to the following rate of change of pressure following a fluid particle:

$$\boxed{\frac{DP}{Dt} = \rho\left[-U_0^{\,2} b - 2U_0 b^2 x + b^3\left(y^2 - x^2\right)\right]} \qquad (3)$$

***Discussion*** The material derivative can be applied to any flow property, scalar or vector. Here we apply it to the pressure, a scalar quantity.

---

## 4-17

**Solution** For a given velocity field we are to calculate the acceleration.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity components are

*Velocity components*: $u = 1.1 + 2.8x + 0.65y \qquad v = 0.98 - 2.1x - 2.8y$ (1)

The acceleration field components are obtained from its definition (the material acceleration) in Cartesian coordinates,

$$\begin{aligned} a_x &= \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = 0 + (1.1 + 2.8x + 0.65y)(2.8) + (0.98 - 2.1x - 2.8y)(0.65) + 0 \\ a_y &= \frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = 0 + (1.1 + 2.8x + 0.65y)(-2.1) + (0.98 - 2.1x - 2.8y)(-2.8) + 0 \end{aligned} \qquad (2)$$

where the unsteady terms are zero since this is a steady flow, and the terms with *w* are zero since the flow is two-dimensional. Eq. 2 simplifies to

*Acceleration components*: $\boxed{a_x = 3.717 + 6.475x \qquad a_y = -5.054 + 6.475y}$ (3)

At the point (*x*,*y*) = (-2,3), the acceleration components of Eq. 3 are

*Acceleration components at* (-2,3): $a_x = -9.233 \cong \mathbf{-9.23} \qquad a_y = 14.371 \cong \mathbf{14.4}$

***Discussion*** The final answers are given to three significant digits. No units are given in either the problem statement or the answers. We assume that the coefficients have appropriate units.

---

## 4-18

**Solution** For a given velocity field we are to calculate the acceleration.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity components are

*Velocity components*: $u = 0.20 + 1.3x + 0.85y \qquad v = -0.50 + 0.95x - 1.3y$ (1)

The acceleration field components are obtained from its definition (the material acceleration) in Cartesian coordinates,

$$\begin{aligned} a_x &= \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = 0 + (0.20 + 1.3x + 0.85y)(1.3) + (-0.50 + 0.95x - 1.3y)(0.85) + 0 \\ a_y &= \frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = 0 + (0.20 + 1.3x + 0.85y)(0.95) + (-0.50 + 0.95x - 1.3y)(-1.3) + 0 \end{aligned} \qquad (2)$$

where the unsteady terms are zero since this is a steady flow, and the terms with *w* are zero since the flow is two-dimensional. Eq. 2 simplifies to

*Acceleration components*: $\boxed{a_x = -0.165 + 2.4975x \qquad a_y = 0.84 + 2.4975y}$ (3)

At the point (*x*,*y*) = (1,2), the acceleration components of Eq. 3 are

*Acceleration components at* (1,2): $a_x = 2.3325 \cong \mathbf{2.33} \qquad a_y = 5.835 \cong \mathbf{5.84}$

***Discussion*** The final answers are given to three significant digits. No units are given in either the problem statement or the answers. We assume that the coefficients have appropriate units.

---

**4-19**

**Solution** We are to generate an expression for the fluid acceleration for a given velocity.

***Assumptions*** **1** The flow is steady. **2** The flow is axisymmetric. **3** The water is incompressible.

***Analysis*** In Problem 4-2 we found that along the centerline,

*Speed along centerline of nozzle*:
$$u = u_{\text{entrance}} + \frac{\left(u_{\text{exit}} - u_{\text{entrance}}\right)}{L^2}x^2 \quad (1)$$

To find the acceleration in the *x*-direction, we use the material acceleration,

*Acceleration along centerline of nozzle*:
$$a_x = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} \quad (2)$$

The first term in Eq. 2 is zero because the flow is steady. The last two terms are zero because the flow is axisymmetric, which means that along the centerline there can be no *v* or *w* velocity component. We substitute Eq. 1 for *u* to obtain

*Acceleration along centerline of nozzle*:
$$a_x = u\frac{\partial u}{\partial x} = \left(u_{\text{entrance}} + \frac{\left(u_{\text{exit}} - u_{\text{entrance}}\right)}{L^2}x^2\right)(2)\frac{\left(u_{\text{exit}} - u_{\text{entrance}}\right)}{L^2}x \quad (3)$$

or

$$\boxed{a_x = 2u_{\text{entrance}}\frac{\left(u_{\text{exit}} - u_{\text{entrance}}\right)}{L^2}x + 2\frac{\left(u_{\text{exit}} - u_{\text{entrance}}\right)^2}{L^4}x^3} \quad (4)$$

***Discussion*** Fluid particles are accelerated along the centerline of the nozzle, even though the flow is steady.

---

**4-20**

**Solution** We are to write an equation for centerline speed through a diffuser, given that the flow speed decreases parabolically.

***Assumptions*** **1** The flow is steady. **2** The flow is axisymmetric.

***Analysis*** A general equation for a parabola in *x* is

*General parabolic equation*:
$$u = a + b\left(x - c\right)^2 \quad (1)$$

We have two boundary conditions, namely at $x = 0$, $u = u_{\text{entrance}}$ and at $x = L$, $u = u_{\text{exit}}$. By inspection, Eq. 1 is satisfied by setting $c = 0$, $a = u_{\text{entrance}}$ and $b = (u_{\text{exit}} - u_{\text{entrance}})/L^2$. Thus, Eq. 1 becomes

*Parabolic speed*:
$$\boxed{u = u_{\text{entrance}} + \frac{\left(u_{\text{exit}} - u_{\text{entrance}}\right)}{L^2}x^2} \quad (2)$$

***Discussion*** You can verify Eq. 2 by plugging in $x = 0$ and $x = L$.

---

**4-21**

**Solution** We are to generate an expression for the fluid acceleration for a given velocity, and then calculate its value at two $x$ locations.

***Assumptions*** **1** The flow is steady. **2** The flow is axisymmetric.

***Analysis*** In the previous problem, we found that along the centerline,

*Speed along centerline of diffuser*:

$$u = u_{\text{entrance}} + \frac{\left(u_{\text{exit}} - u_{\text{entrance}}\right)}{L^2}x^2 \qquad (1)$$

To find the acceleration in the $x$-direction, we use the material acceleration,

*Acceleration along centerline of diffuser*:

$$a_x = \cancel{\frac{\partial u}{\partial t}} + u\frac{\partial u}{\partial x} + \cancel{v\frac{\partial u}{\partial y}} + \cancel{w\frac{\partial u}{\partial z}} \qquad (2)$$

The first term in Eq. 2 is zero because the flow is steady. The last two terms are zero because the flow is axisymmetric, which means that along the centerline there can be no $v$ or $w$ velocity component. We substitute Eq. 1 for $u$ to obtain

*Acceleration along centerline of diffuser*:

$$a_x = u\frac{\partial u}{\partial x} = \left(u_{\text{entrance}} + \frac{\left(u_{\text{exit}} - u_{\text{entrance}}\right)}{L^2}x^2\right)(2)\frac{\left(u_{\text{exit}} - u_{\text{entrance}}\right)}{L^2}x$$

or

$$\boxed{a_x = 2u_{\text{entrance}}\frac{\left(u_{\text{exit}} - u_{\text{entrance}}\right)}{L^2}x + 2\frac{\left(u_{\text{exit}} - u_{\text{entrance}}\right)^2}{L^4}x^3} \qquad (3)$$

At the given locations, we substitute the given values. At $x = 0$,

*Acceleration along centerline of diffuser at $x = 0$*:

$$a_x\left(x = 0\right) = \mathbf{0} \qquad (4)$$

At $x = 1.0$ m,

*Acceleration along centerline of diffuser at $x = 1.0$ m*:

$$a_x\left(x = 1.0\text{ m}\right) = 2\left(30.0\text{ m/s}\right)\frac{\left(-25.0\text{ m/s}\right)}{\left(2.0\text{ m}\right)^2}\left(1.0\text{ m}\right) + 2\frac{\left(-25.0\text{ m/s}\right)^2}{\left(2.0\text{ m}\right)^4}\left(1.0\text{ m}\right)^3 \qquad (5)$$

$$= \mathbf{-297\ m/s^2}$$

***Discussion*** $a_x$ is negative implying that fluid particles are *decelerated* along the centerline of the diffuser, even though the flow is steady. Because of the parabolic nature of the velocity field, the acceleration is zero at the entrance of the diffuser, but its magnitude increases rapidly downstream.

## Flow Patterns and Flow Visualization

### 4-22C

**Solution** We are to define streamline and discuss what streamlines indicate.

***Analysis*** A *streamline* is a **curve that is everywhere tangent to the instantaneous local velocity vector**. It indicates the instantaneous direction of fluid motion throughout the flow field.

***Discussion*** If a flow field is steady, streamlines, pathlines, and streaklines are identical.

### 4-23

**Solution** For a given velocity field we are to generate an equation for the streamlines.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The steady, two-dimensional velocity field of Problem 4-15 is

*Velocity field*:
$$\vec{V} = (u, v) = (U_0 + bx)\vec{i} - by\vec{j} \qquad (1)$$

For two-dimensional flow in the *x*-*y* plane, streamlines are given by

*Streamlines in the x-y plane*:
$$\left.\frac{dy}{dx}\right)_{\text{along a streamline}} = \frac{v}{u} \qquad (2)$$

We substitute the *u* and *v* components of Eq. 1 into Eq. 2 and rearrange to get

$$\frac{dy}{dx} = \frac{-by}{U_0 + bx}$$

We solve the above differential equation by separation of variables:

$$-\int \frac{dy}{by} = \int \frac{dx}{U_0 + bx}$$

Integration yields

$$-\frac{1}{b}\ln(by) = \frac{1}{b}\ln(U_0 + bx) + \frac{1}{b}\ln C_1 \qquad (3)$$

where we have set the constant of integration as the natural logarithm of some constant $C_1$, with a constant in front in order to simplify the algebra (notice that the factor of 1/*b* can be removed from each term in Eq. 3). When we recall that ln(*ab*) = ln*a* + ln*b*, and that –ln*a* = ln(1/*a*), Eq. 3 simplifies to

*Equation for streamlines*:
$$\boxed{y = \frac{C}{(U_0 + bx)}} \qquad (4)$$

The new constant *C* is related to $C_1$, and is introduced for simplicity.

***Discussion*** Each value of constant *C* yields a unique streamline of the flow.

## 4-24E

**Solution** For a given velocity field we are to plot several streamlines for a given range of $x$ and $y$ values.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $x$-$y$ plane.

***Analysis*** From the solution to the previous problem, an equation for the streamlines is

*Streamlines in the x-y plane*:
$$y = \frac{C}{\left(U_0 + bx\right)} \qquad (1)$$

Constant $C$ is set to various values in order to plot the streamlines. Several streamlines in the given range of $x$ and $y$ are plotted in **Fig. 1**.

The direction of the flow is found by calculating $u$ and $v$ at some point in the flow field. We choose $x$ = 1 ft, $y$ = 1 ft. At this point $u$ = 9.6 ft/s and $v$ = –4.6 ft/s. The direction of the velocity at this point is obviously to the lower right. This sets the direction of all the streamlines. The arrows in Fig. 1 indicate the direction of flow.

***Discussion*** The flow is type of converging channel flow.

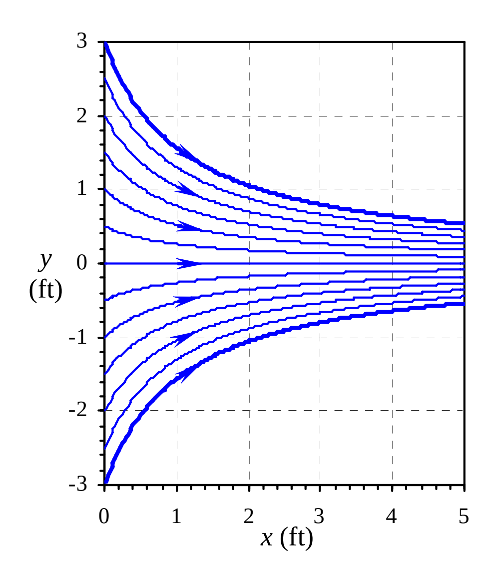



**FIGURE 1**
Streamlines (solid blue curves) for the given velocity field; $x$ and $y$ are in units of ft.

---

## 4-25C

**Solution** We are to determine what kind of flow visualization is seen in a photograph.

***Analysis*** Since the picture is a snapshot of dye streaks in water, each streak shows the time history of dye that was introduced earlier from a port in the body. Thus these are **streaklines**. Since the flow appears to be steady, these streaklines are the same as pathlines and streamlines.

***Discussion*** It is assumed that the dye follows the flow of the water. If the dye is of nearly the same density as the water, this is a reasonable assumption.

---

## 4-26C

**Solution** We are to define pathline and discuss what pathlines indicate.

***Analysis*** A *pathline* is the **actual path traveled by an individual fluid particle over some time period**. It indicates the exact route along which a fluid particle travels from its starting point to its ending point. Unlike streamlines, pathlines are not instantaneous, but involve a finite time period.

***Discussion*** If a flow field is steady, streamlines, pathlines, and streaklines are identical.

---

## 4-27C

**Solution** We are to define streakline and discuss the difference between streaklines and streamlines.

***Analysis*** A *streakline* is the **locus of fluid particles that have passed sequentially through a prescribed point in the flow**. Streaklines are very different than streamlines. Streamlines are instantaneous curves, everywhere tangent to the local velocity, while streaklines are produced over a finite time period. In an unsteady flow, streaklines distort and then retain features of that distorted shape even as the flow field changes, whereas streamlines change instantaneously with the flow field.

***Discussion*** If a flow field is steady, streamlines and streaklines are identical.

---

**4-28C**

**Solution** We are to determine what kind of flow visualization is seen in a photograph.

***Analysis*** Since the picture is a snapshot of dye streaks in water, each streak shows the time history of dye that was introduced earlier from a port in the body. Thus these are **streaklines**. Since the flow appears to be unsteady, these streaklines are *not* the same as pathlines or streamlines.

***Discussion*** It is assumed that the dye follows the flow of the water. If the dye is of nearly the same density as the water, this is a reasonable assumption.

---

**4-29C**

**Solution** We are to determine what kind of flow visualization is seen in a photograph.

***Analysis*** Since the picture is a snapshot of smoke streaks in air, each streak shows the time history of smoke that was introduced earlier from the smoke wire. Thus these are **streaklines**. Since the flow appears to be unsteady, these streaklines are *not* the same as pathlines or streamlines.

***Discussion*** It is assumed that the smoke follows the flow of the air. If the smoke is neutrally buoyant, this is a reasonable assumption. In actuality, the smoke rises a bit since it is hot; however, the air speeds are high enough that this effect is negligible.

---

**4-30C**

**Solution** We are to determine what kind of flow visualization is seen in a photograph.

***Analysis*** Since the picture is a time exposure of air bubbles in water, each white streak shows the path of an individual air bubble. Thus these are **pathlines**. Since the outer flow (top and bottom portions of the photograph) appears to be steady, these pathlines are the same as streaklines and streamlines.

***Discussion*** It is assumed that the air bubbles follow the flow of the water. If the bubbles are small enough, this is a reasonable assumption.

---

**4-31C**

**Solution** We are to define timeline and discuss how timelines can be produced in a water channel. We are also to describe an application where timelines are more useful than streaklines.

***Analysis*** A *timeline* is a **set of adjacent fluid particles that were marked at the same instant of time**. Timelines can be produced in a water flow by using a hydrogen bubble wire. There are also techniques in which a chemical reaction is initiated by applying current to the wire, changing the fluid color along the wire. Timelines are more useful than streaklines when the uniformity of a flow is to be visualized. Another application is to visualize the velocity profile of a boundary layer or a channel flow.

***Discussion*** Timelines differ from streamlines, streaklines, and pathlines even if the flow is steady.

---

**4-32C**

**Solution** For each case we are to decide whether a vector plot or contour plot is most appropriate, and we are to explain our choice.

***Analysis*** In general, contour plots are most appropriate for scalars, while vector plots are necessary when vectors are to be visualized.

(*a*) A **contour plot** of speed is most appropriate since fluid speed is a scalar.
(*b*) A **vector plot** of velocity vectors would clearly show where the flow separates. Alternatively, a vorticity contour plot of vorticity normal to the plane would also show the separation region clearly.
(*c*) A **contour plot** of temperature is most appropriate since temperature is a scalar.
(*d*) A **contour plot** of this component of vorticity is most appropriate since one component of a vector is a scalar.

***Discussion*** There are other options for case (b) – temperature contours can also sometimes be used to identify a separation zone.

---

**4-33**

**Solution** For a given velocity field we are to generate an equation for the streamlines and sketch several streamlines in the first quadrant.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity field is given by

$$\vec{V} = (u, v) = (0.5 + 1.2x)\vec{i} + (-2.0 - 1.2y)\vec{j} \qquad (1)$$

For two-dimensional flow in the *x*-*y* plane, streamlines are given by

*Streamlines in the x-y plane*:
$$\left.\frac{dy}{dx}\right)_{\text{along a streamline}} = \frac{v}{u} \qquad (2)$$

We substitute the *u* and *v* components of Eq. 1 into Eq. 2 and rearrange to get

$$\frac{dy}{dx} = \frac{-2.0 - 1.2y}{0.5 + 1.2x}$$

We solve the above differential equation by separation of variables:

$$\frac{dy}{-2.0 - 1.2y} = \frac{dx}{0.5 + 1.2x} \quad \rightarrow \quad \int \frac{dy}{-2.0 - 1.2y} = \int \frac{dx}{0.5 + 1.2x}$$

Integration yields

$$-\frac{1}{1.2}\ln(-2.0 - 1.2y) = \frac{1}{1.2}\ln(0.5 + 1.2x) - \frac{1}{1.2}\ln C_1 \qquad (3)$$

where we have set the constant of integration as the natural logarithm of some constant $C_1$, with a constant in front in order to simplify the algebra. When we recall that ln(*ab*) = ln*a* + ln*b*, and that –ln*a* = ln(1/*a*), Eq. 3 simplifies to

*Equation for streamlines*:
$$\boxed{y = \frac{C}{1.2(0.5 + 1.2x)} - 1.667}$$

The new constant *C* is related to $C_1$, and is introduced for simplicity. *C* can be set to various values in order to plot the streamlines. Several streamlines in the upper right quadrant of the given flow field are shown in **Fig. 1**.

The direction of the flow is found by calculating *u* and *v* at some point in the flow field. We choose *x* = 3, *y* = 3. At this point *u* = 4.1 and *v* = -5.6. The direction of the velocity at this point is obviously to the lower right. This sets the direction of all the streamlines. The arrows in Fig. 1 indicate the direction of flow.

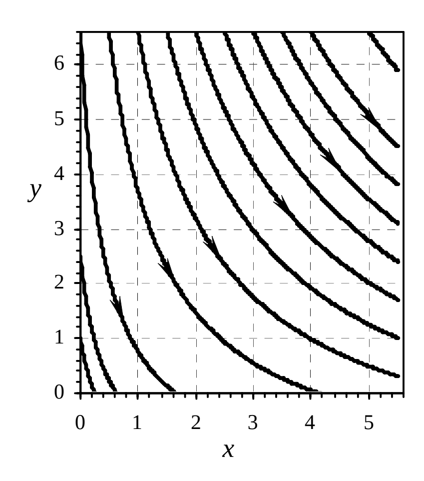



**FIGURE 1**
Streamlines (solid black curves) for the given velocity field.

***Discussion*** The flow appears to be a counterclockwise turning flow in the upper right quadrant.

**4-34**

**Solution** For a given velocity field we are to generate a velocity vector plot in the first quadrant.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity field is given by

$$\vec{V} = (u,v) = (0.5 + 1.2x)\vec{i} + (-2.0 - 1.2y)\vec{j} \qquad (1)$$

At any point (*x*,*y*) in the flow field, the velocity components *u* and *v* are obtained from Eq. 1,

*Velocity components*: $u = 0.5 + 1.2x \qquad v = -2.0 - 1.2y \qquad (2)$

To plot velocity vectors, we simply pick an (*x*,*y*) point, calculate *u* and *v* from Eq. 2, and plot an arrow with its tail at (*x*,*y*), and its tip at (*x*+*Su*,*y*+*Sv*) where *S* is some scale factor for the vector plot. For the vector plot shown in **Fig. 1**, we chose *S* = 0.2, and plot velocity vectors at several locations in the first quadrant.

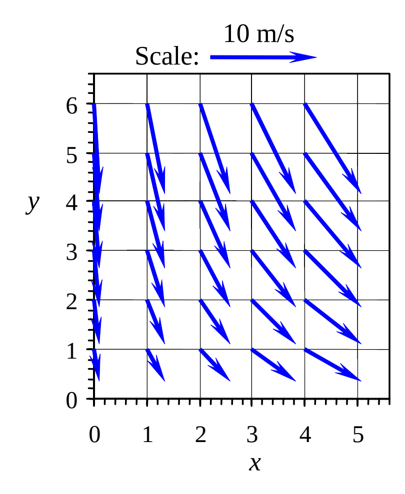



**FIGURE 1**
Velocity vectors for the given velocity field. The scale is shown by the top arrow.

***Discussion*** The flow appears to be a counterclockwise turning flow in the upper right quadrant.

**4-35**

**Solution** For a given velocity field we are to generate an acceleration vector plot in the first quadrant.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity field is given by

$$\vec{V} = (u,v) = (0.5 + 1.2x)\vec{i} + (-2.0 - 1.2y)\vec{j} \qquad (1)$$

At any point (*x*,*y*) in the flow field, the velocity components *u* and *v* are obtained from Eq. 1,

*Velocity components*: $u = 0.5 + 1.2x \qquad v = -2.0 - 1.2y \qquad (2)$

The acceleration field is obtained from its definition (the material acceleration),

*Acceleration components*:

$$a_x = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = 0 + (0.5 + 1.2x)(1.2) + 0 + 0$$
$$a_y = \frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = 0 + 0 + (-2.0 - 1.2y)(-1.2) + 0 \qquad (3)$$

where the unsteady terms are zero since this is a steady flow, and the terms with *w* are zero since the flow is two-dimensional. Eq. 3 simplifies to

*Acceleration components*: $a_x = 0.6 + 1.44x \qquad a_y = 2.4 + 1.44y \qquad (4)$

To plot the acceleration vectors, we simply pick an (*x*,*y*) point, calculate $a_x$ and $a_y$ from Eq. 4, and plot an arrow with its tail at (*x*,*y*), and its tip at ($x+Sa_x$,$y+Sa_y$) where *S* is some scale factor for the vector plot. For the vector plot shown in **Fig. 1**, we chose *S* = 0.15, and plot acceleration vectors at several locations in the first quadrant.

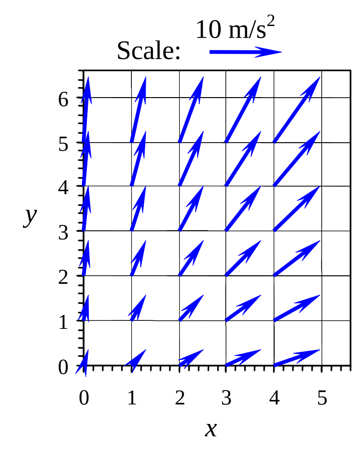



**FIGURE 1**
Acceleration vectors for the velocity field. The scale is shown by the top arrow.

***Discussion*** Since the flow is a counterclockwise turning flow in the upper right quadrant, the acceleration vectors point to the upper right (centripetal acceleration).

**4-36**

**Solution** For the given velocity field, the location(s) of stagnation point(s) are to be determined. Several velocity vectors are to be sketched and the velocity field is to be described.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is two-dimensional, implying no $z$-component of velocity and no variation of $u$ or $v$ with $z$.

***Analysis*** ($a$) The velocity field is

$$\vec{V} = (u, v) = (1 + 2.5x + y)\vec{i} + (-0.5 - 1.5x - 2.5y)\vec{j} \qquad (1)$$

Since $\vec{V}$ is a vector, all its components must equal zero in order for $\vec{V}$ itself to be zero. Setting each component of Eq. 1 to zero,

*Simultaneous equations*:
$$u = \quad 1 + 2.5x + \quad y = 0$$
$$v = -0.5 - 1.5x - 2.5y = 0$$

We can easily solve this set of two equations and two unknowns simultaneously. Yes, there is one stagnation point, and it is located at

*Stagnation point*: $x =$ **-0.421 m** $\qquad y =$ **0.0526 m**

($b$) The $x$ and $y$ components of velocity are calculated from Eq. 1 for several ($x$,$y$) locations in the specified range. For example, at the point ($x$ = 2 m, $y$ = 3 m), $u$ = 9.00 m/s and $v$ = -11 m/s. The magnitude of velocity (the *speed*) at that point is 14.21 m/s. At this and at an array of other locations, the velocity vector is constructed from its two components, the results of which are shown in Fig. 1. The flow can be described as a counterclockwise turning, accelerating flow from the upper left to the lower right. The stagnation point of Part ($a$) does not lie in the upper right quadrant, and therefore does not appear on the sketch.

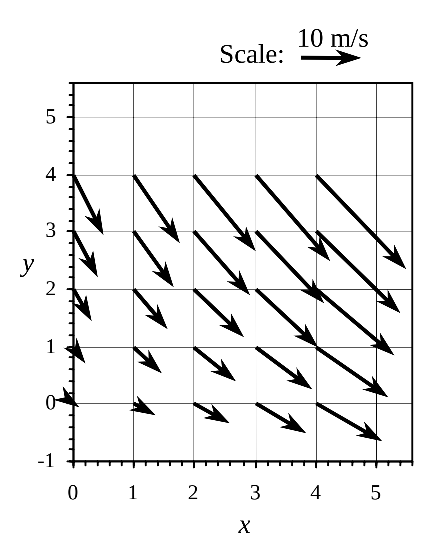



**FIGURE 1**
Velocity vectors in the upper right quadrant for the given velocity field.

***Discussion*** The stagnation point location is given to three significant digits. It will be verified in Chap. 9 that this flow field is physically valid because it satisfies the differential equation for conservation of mass.

---

**4-37**

**Solution** For the given velocity field, the material acceleration is to be calculated at a particular point and plotted at several locations in the upper right quadrant.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is two-dimensional, implying no $z$-component of velocity and no variation of $u$ or $v$ with $z$.

***Analysis*** ($a$) The velocity field is

$$\vec{V} = (u, v) = (1 + 2.5x + y)\vec{i} + (-0.5 - 1.5x - 2.5y)\vec{j} \qquad (1)$$

Using the velocity field of Eq. 1 and the equation for material acceleration in Cartesian coordinates, we write expressions for the two non-zero components of the acceleration vector:

$$a_x = \frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}$$
$$= 0 + (1 + 2.5x + y)(2.5) + (-0.5 - 1.5x - 2.5y)(1) + 0$$

and

$$a_y = \frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}$$
$$= 0 + (1 + 2.5x + y)(-1.5) + (-0.5 - 1.5x - 2.5y)(-2.5) + 0$$

At ($x$ = 2 m, $y$ = 3 m), $a_x$ = **11.5 m/s²** and $a_y$ = **14.0 m/s²**.

($b$) The above equations are applied to an array of $x$ and $y$ values in the upper right quadrant, and the acceleration vectors are plotted in Fig. 1.

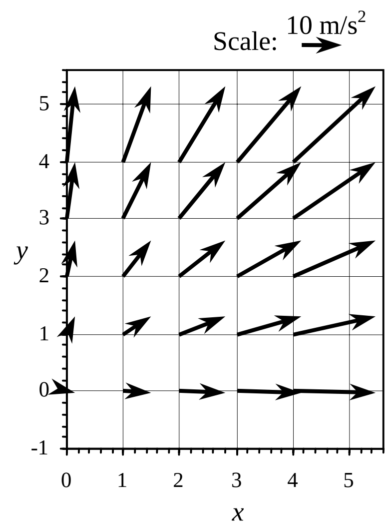



**FIGURE 1**
Acceleration vectors in the upper right quadrant for the given velocity field.

***Discussion*** The acceleration vectors plotted in Fig. 1 point to the upper right, increasing in magnitude away from the origin. This agrees qualitatively with the velocity vectors of Fig. 1 of the previous problem; namely, fluid particles are accelerated to the right and are turned in the counterclockwise direction due to centripetal acceleration towards the upper right. Note that the acceleration field is non-zero, even though the flow is steady.

**4-38**

**Solution** For a given velocity field we are to plot a velocity magnitude contour plot at five given values of speed.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis*** Since $u_r = 0$, and since $\omega$ is positive, the speed is equal to the magnitude of the $\theta$-component of velocity,

*Speed*:
$$V = \sqrt{\underbrace{u_r^2}_{0} + u_\theta^{\,2}} = |u_\theta| = \omega r$$

Thus, contour lines of constant speed are simply circles of constant radius given by

*Contour line of constant speed*:
$$r = \frac{V}{\omega}$$

For example, at $V$ = 2.0 m/s, the corresponding contour line is a circle of radius 2.0 m,

*Contour line at constant speed V = 2.0 m/s*:
$$r = \frac{2.0 \text{ m/s}}{1.0 \text{ 1/s}} = 2.0 \text{ m}$$

We plot a circle at a radius of 2.0 m and repeat this simple calculation for the four other values of $V$. We plot the contours in **Fig. 1**. The speed increases linearly from the center of rotation (the origin).

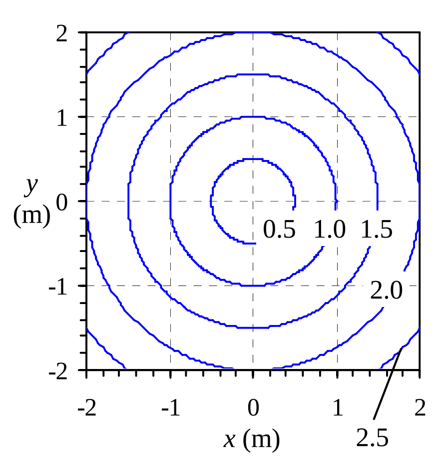



**FIGURE 1**
Contour plot of velocity magnitude for solid body rotation. Values of speed are labeled in units of m/s.

***Discussion*** The contours are equidistant apart because of the linear nature of the velocity field.

**4-39**

**Solution** For a given velocity field we are to plot a velocity magnitude contour plot at five given values of speed.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis*** Since $u_r = 0$, and since $K$ is positive, the speed is equal to the magnitude of the $\theta$-component of velocity,

*Speed*:
$$V = \sqrt{\underbrace{u_r^2}_{0} + u_\theta^{\,2}} = |u_\theta| = \frac{K}{r}$$

Thus, contour lines of constant speed are simply circles of constant radius given by

*Contour line of constant speed*:
$$r = \frac{K}{V}$$

For example, at $V$ = 2.0 m/s, the corresponding contour line is a circle of radius 0.50 m,

*Contour line at constant speed V = 2.0 m/s*:
$$r = \frac{1.0 \text{ m}^2\text{/s}}{2.0 \text{ m/s}} = 0.50 \text{ m}$$

We plot a circle at a radius of 0.50 m and repeat this simple calculation for the four other values of $V$. We plot the contours in **Fig. 1**. The speed near the center is faster than that further away from the center.

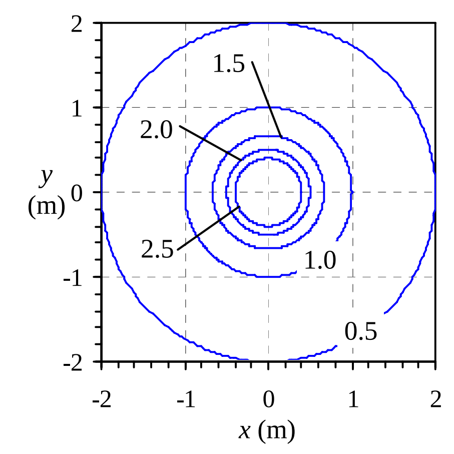



**FIGURE 1**
Contour plot of velocity magnitude for a line vortex. Values of speed are labeled in units of m/s.

***Discussion*** The contours are *not* equidistant apart because of the nonlinear nature of the velocity field.

**4-40**

**Solution** For a given velocity field we are to plot a velocity magnitude contour plot at five given values of speed.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis*** The velocity field is

*Line source*:
$$u_r = \frac{m}{2\pi r} \qquad u_\theta = 0 \qquad (1)$$

Since $u_\theta = 0$, and since $m$ is positive, the speed is equal to the magnitude of the $r$-component of velocity,

*Speed*:
$$V = \sqrt{u_r^{\,2} + \underbrace{u_\theta^{\,2}}_{0}} = |u_r| = \frac{m}{2\pi r} \qquad (2)$$

Thus, contour lines of constant speed are simply circles of constant radius given by

*Contour line of constant speed*:
$$r = \frac{m}{2\pi V} = \frac{\left(\frac{m}{2\pi}\right)}{V} \qquad (3)$$

For example, at $V$ = 2.0 m/s, the corresponding contour line is a circle of radius 0.50 m,

*Contour line at speed V = 2.0 m/s*:
$$r = \frac{1.0 \text{ m}^2\text{/s}}{2.0 \text{ m/s}} = 0.50 \text{ m} \qquad (4)$$

We plot a circle at a radius of 0.50 m and repeat this simple calculation for the four other values of $V$. We plot the contours in **Fig. 1**. The flow slows down as it travels further from the origin.

***Discussion*** The contours are *not* equidistant apart because of the nonlinear nature of the velocity field.

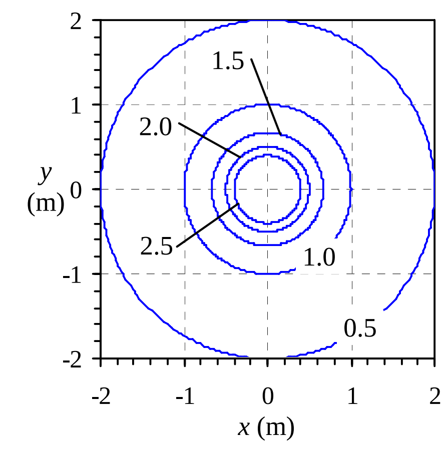



**FIGURE 1**
Contour plot of velocity magnitude for a line source. Values of speed are labeled in units of m/s.

## Motion and Deformation of Fluid Elements

### 4-41C

**Solution** We are to name and describe the four fundamental types of motion or deformation of fluid particles.

***Analysis***

1. **Translation** – a fluid particle **moves from one location to another**.
2. **Rotation** – a fluid particle **rotates about an axis** drawn through the particle.
3. **Linear strain or extensional strain** – a fluid particle **stretches in a direction such that a line segment in that direction is elongated at some later time**.
4. **Shear strain** – a fluid particle distorts in such a way that **two lines through the fluid particle that are initially perpendicular are *not* perpendicular at some later time**.

***Discussion*** In a complex fluid flow, all four of these occur simultaneously.

### 4-42

**Solution** For a given velocity field, we are to determine whether the flow is rotational or irrotational.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity field is

$$\vec{V} = (u, v) = (U_0 + bx)\vec{i} - by\vec{j} \qquad (1)$$

By definition, the flow is rotational if the vorticity is non-zero. So, we calculate the vorticity. In a 2-D flow in the *x*-*y* plane, the only non-zero component of vorticity is in the *z* direction, i.e. $\zeta_z$,

*Vorticity component in the z direction*:

$$\zeta_z = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 - 0 = 0 \qquad (1)$$

Since the vorticity is zero, this flow is **irrotational**.

***Discussion*** We shall see in Chap. 10 that the fluid very close to the walls is rotational due to important viscous effects near the wall (a *boundary layer*). However, in the majority of the flow field, the irrotational approximation is reasonable.

**4-43**

**Solution** For a given velocity field we are to generate an equation for the *x* location of a fluid particle along the *x*-axis as a function of time.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity field is

*Velocity field*:
$$\vec{V} = (u, v) = (U_0 + bx)\vec{i} - by\vec{j} \qquad (1)$$

We start with the definition of *u* following a fluid particle,

*x-component of velocity of a fluid particle*:
$$\frac{dx_{\text{particle}}}{dt} = u = U_0 + bx_{\text{particle}} \qquad (2)$$

where we have substituted *u* from Eq. 1. We rearrange and separate variables, dropping the "particle" subscript for convenience,

$$\frac{dx}{U_0 + bx} = dt \qquad (3)$$

Integration yields

$$\frac{1}{b}\ln(U_0 + bx) = t - \frac{1}{b}\ln C_1 \qquad (4)$$

where we have set the constant of integration as the natural logarithm of some constant $C_1$, with a constant in front in order to simplify the algebra. When we recall that ln(*ab*) = ln*a* + ln*b*, Eq. 4 simplifies to

$$\ln\left(C_1\left(U_0 + bx\right)\right) = t$$

from which

$$U_0 + bx = C_2 e^{bt} \qquad (5)$$

where $C_2$ is a new constant defined for convenience. We now plug in the known initial condition that at $t = 0$, $x = x_A$ to find constant $C_2$ in Eq. 5. After some algebra,

*Fluid particle's x location at time t*:
$$\boxed{x = x_{A'} = \frac{1}{b}\left[\left(U_0 + bx_A\right)e^{bt} - U_0\right]} \qquad (6)$$

***Discussion*** We verify that at $t = 0$, $x = x_A$ in Eq. 6.

**4-44**

**Solution** For a given velocity field we are to generate an equation for the change in length of a line segment moving with the flow along the *x*-axis.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** Using the results of the previous problem,

*Location of particle A at time t*: $$x_{A'} = \frac{1}{b}\left[\left(U_0 + bx_B\right)e^{bt} - U_0\right] \qquad (1)$$

and

*Location of particle B at time t*: $$x_{B'} = \frac{1}{b}\left[\left(U_0 + bx_B\right)e^{bt} - U_0\right] \qquad (2)$$

Since length $\xi = x_B - x_A$ and length $\xi + \Delta\xi = x_{B'} - x_{A'}$, we write an expression for $\Delta\xi$,

*Change in length of the line segment*:

$$\begin{aligned}\Delta\xi &= \left(x_{B'} - x_{A'}\right) - \left(x_B - x_A\right)\\ &= \frac{1}{b}\left[\left(U_0 + bx_B\right)e^{bt} - U_0\right] - \frac{1}{b}\left[\left(U_0 + bx_A\right)e^{bt} - U_0\right] - \left(x_B - x_A\right) \qquad (3)\\ &= x_B e^{bt} - x_A e^{bt} - x_B + x_A\end{aligned}$$

Eq. 3 simplifies to

*Change in length of the line segment*: $$\boxed{\Delta\xi = \left(x_B - x_A\right)\left(e^{bt} - 1\right)} \qquad (4)$$

***Discussion*** We verify from Eq. 4 that when $t = 0$, $\Delta\xi = 0$.

**4-45**

**Solution** By examining the increase in length of a line segment along the axis of a converging duct, we are to generate an equation for linear strain rate in the *x* direction and compare to the exact equation given in this chapter.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** From the previous problem, we have an expression for the change in length of the line segment AB,

*Change in length of the line segment*: $$\Delta\xi = \left(x_B - x_A\right)\left(e^{bt} - 1\right) \qquad (1)$$

The fundamental definition of linear strain rate is the rate of increase in length of a line segment per unit length of the line segment. For the case at hand,

*Linear strain rate in x direction*: $$\varepsilon_{xx} = \frac{d}{dt}\frac{\left(\xi + \Delta\xi\right) - \xi}{\xi} = \frac{d}{dt}\frac{\Delta\xi}{\xi} = \frac{d}{dt}\frac{\Delta\xi}{x_B - x_A} \qquad (2)$$

We substitute Eq. 1 into Eq. 2 to obtain

*Linear strain rate in x direction*: $$\varepsilon_{xx} = \frac{d}{dt}\frac{\left(x_B - x_A\right)\left(e^{bt} - 1\right)}{x_B - x_A} = \frac{d}{dt}\left(e^{bt} - 1\right) \qquad (3)$$

In the limit as $t \to 0$, we apply the first two terms of the series expansion for $e^{bt}$,

*Series expansion for* $e^{bt}$: $$e^{bt} = 1 + bt + \frac{\left(bt\right)^2}{2!} + ... \approx 1 + bt \qquad (4)$$

Finally, for small *t* we approximate the time derivative as $1/t$, yielding

*Linear strain rate in x direction*: $$\boxed{\varepsilon_{xx} \to \frac{1}{t}\left(1 + bt - 1\right) = b} \qquad (5)$$

Comparing to the equation for $\varepsilon_{xx}$,

*Linear strain rate in x direction*: $$\boxed{\varepsilon_{xx} = \frac{\partial u}{\partial x} = b} \qquad (6)$$

Equations 5 and 6 agree, verifying our algebra.

***Discussion*** Although we considered a line segment on the *x*-axis, it turns out that $\varepsilon_{xx} = b$ everywhere in the flow, as seen from Eq. 6. We could also have taken the analytical time derivative of Eq. 3, yielding $\varepsilon_{xx} = be^{bt}$. Then, as $t \to 0$, $\varepsilon_{xx} \to b$.

**4-46**

**Solution** For a given velocity field we are to generate an equation for the *y* location of a fluid particle as a function of time.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity field is

*Velocity field*: $$\vec{V} = (u, v) = (U_0 + bx)\vec{i} - by\vec{j} \qquad (1)$$

We start with the definition of *v* following a fluid particle,

*y-component of velocity of a fluid particle*: $$\frac{dy_{\text{particle}}}{dt} = v = -by_{\text{particle}} \qquad (2)$$

where we have substituted *v* from Eq. 1. We and rearrange and separate variables, dropping the "particle" subscript for convenience,

$$\frac{dy}{y} = -bdt \qquad (3)$$

Integration yields

$$\ln(y) = -bt - \ln C_1 \qquad (4)$$

where we have set the constant of integration as the natural logarithm of some constant $C_1$, with a constant in front in order to simplify the algebra. When we recall that ln(*ab*) = ln*a* + ln*b*, Eq. 4 simplifies to

$$\ln(C_1 y) = -t$$

from which

$$y = C_2 e^{-bt} \qquad (5)$$

where $C_2$ is a new constant defined for convenience. We now plug in the known initial condition that at $t = 0$, $y = y_A$ to find constant $C_2$ in Eq. 5. After some algebra,

*Fluid particle's y location at time t*: $$\boxed{y = y_{A'} = y_A e^{-bt}} \qquad (6)$$

***Discussion*** The fluid particle approaches the *x*-axis exponentially with time. The fluid particle also moves downstream in the *x* direction during this time period. However, in this particular problem *v* is not a function of *x*, so the streamwise movement is irrelevant (*u* and *v* act independently of each other).

**4-47**

**Solution** For a given velocity field we are to generate an equation for the change in length of a line segment in the $y$ direction.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $x$-$y$ plane.

***Analysis*** Using the results of the previous problem,

*Location of particle A at time t*: $$y_{A'} = y_A e^{-bt} \qquad (1)$$

and

*Location of particle B at time t*: $$y_{B'} = y_B e^{-bt} \qquad (2)$$

Since length $\eta = y_B - y_A$ and length $\eta + \Delta\eta = y_{B'} - y_{A'}$, we write an expression for $\Delta\eta$,

*Change in length of the line segment*:

$$\Delta\eta = \left(y_{B'} - y_{A'}\right) - \left(y_B - y_A\right) = y_B e^{-bt} - y_A e^{-bt} - \left(y_B - y_A\right) = y_B e^{-bt} - y_A e^{-bt} - y_B + y_A$$

which simplifies to

*Change in length of the line segment*: $$\boxed{\Delta\eta = \left(y_B - y_A\right)\left(e^{-bt} - 1\right)} \qquad (3)$$

***Discussion*** We verify from Eq. 3 that when $t = 0$, $\Delta\eta = 0$.

---

**4-48**

**Solution** By examining the increase in length of a line segment as it moves down a converging duct, we are to generate an equation for linear strain rate in the *y* direction and compare to the exact equation given in this chapter.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** From the previous problem we have an expression for the change in length of the line segment AB,

*Change in length of the line segment*: $$\Delta\eta = \left(y_{\mathrm{B}} - y_{\mathrm{A}}\right)\left(e^{-bt} - 1\right) \qquad (1)$$

The fundamental definition of linear strain rate is the rate of increase in length of a line segment per unit length of the line segment. For the case at hand,

*Linear strain rate in y direction*:

$$\varepsilon_{yy} = \frac{d}{dt}\frac{\left(\eta + \Delta\eta\right) - \eta}{\eta} = \frac{d}{dt}\frac{\Delta\eta}{\eta} = \frac{d}{dt}\frac{\Delta\eta}{y_{\mathrm{B}} - y_{\mathrm{A}}} \qquad (2)$$

We substitute Eq. 1 into Eq. 2 to obtain

*Linear strain rate in y direction*: $$\varepsilon_{yy} = \frac{d}{dt}\frac{\left(y_{\mathrm{B}} - y_{\mathrm{A}}\right)\left(e^{-bt} - 1\right)}{y_{\mathrm{B}} - y_{\mathrm{A}}} = \frac{d}{dt}\left(e^{-bt} - 1\right) \qquad (3)$$

In the limit as $t \to 0$, we apply the first two terms of the series expansion for $e^{-bt}$,

*Series expansion for $e^{-bt}$*: $$e^{-bt} = 1 + \left(-bt\right) + \frac{\left(-bt\right)^2}{2!} + \ldots \approx 1 - bt \qquad (4)$$

Finally, for small *t* we approximate the time derivative as $1/t$, yielding

*Linear strain rate in y direction*: $$\boxed{\varepsilon_{yy} \to \frac{1}{t}\left(1 - bt - 1\right) = -b} \qquad (5)$$

Comparing to the equation for $\varepsilon$,

*Linear strain rate in y direction*: $$\boxed{\varepsilon_{yy} = \frac{\partial v}{\partial y} = -b} \qquad (6)$$

Equations 5 and 6 agree, verifying our algebra.

***Discussion*** Since *v* does not depend on *x* location in this particular problem, the algebra is simple. In a more general case, both *u* and *v* depend on both *x* and *y*, and a numerical integration scheme is required. We could also have taken the analytical time derivative of Eq. 3, yielding $\varepsilon_{yy} = -be^{-bt}$. Then, as $t \to 0$, $\varepsilon_{xx} \to -b$.



## 4-49E



**Solution** For a given velocity field and an initially square fluid particle, we are to calculate and plot its location and shape after a given time period.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** Using the results of Problems 4-43 and 4-46, we can calculate the location of any point on the fluid particle after the elapsed time. We pick 6 points along each edge of the fluid particle, and plot their *x* and *y* locations at $t = 0$ and at $t = 0.2$ s. For example, the point at the lower left corner of the particle is initially at $x = 0.25$ ft and $y = 0.75$ ft at $t = 0$. At $t = 0.2$ s,

*x-location of lower left corner of the fluid particle at time t = 0.2 s*:

$$x = \frac{1}{4.6\ 1/\text{s}}\left[\left(5.0\ \text{ft/s} + \left(4.6\ 1/\text{s}\right)\left(0.25\ \text{ft}\right)\right)e^{(4.6\ 1/\text{s})(0.2\ \text{s})} - 5.0\ \text{ft/s}\right] = \mathbf{2.268\ ft}$$

and

*y-location of lower left corner of the fluid particle at time t = 0.2 s*:

$$y = \left(0.75\ \text{ft}\right)e^{-(4.6\ 1/\text{s})(0.2\ \text{s})} = \mathbf{0.2989\ ft}$$

We repeat the above calculations at all the points along the edges of the fluid particle, and plot both their initial and final positions in **Fig. 1** as dots. Finally, we connect the dots to draw the fluid particle shape. It is clear from the results that the fluid particle shrinks in the *y* direction and stretches in the *x* direction. However, it does not shear or rotate.

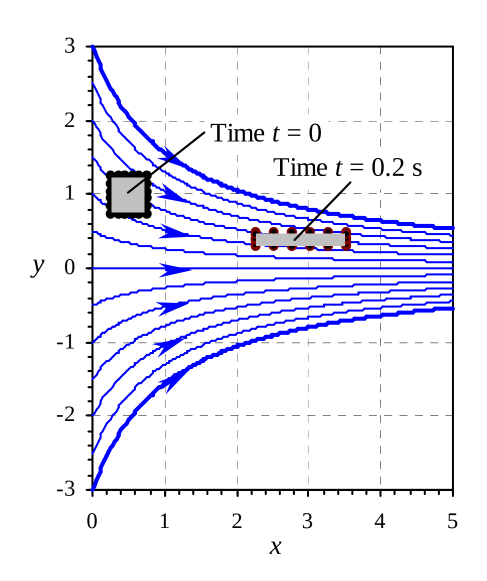



**FIGURE 1**
Movement and distortion of an initially square fluid particle in a converging duct; *x* and *y* are in units of ft. Streamlines (solid blue curves) are also shown for reference.

***Discussion*** The flow is irrotational since fluid particles do not rotate.

## 4-50E

**Solution** By analyzing the shape of a fluid particle, we are to verify that the given flow field is incompressible.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** Since the flow is two-dimensional, we assume unit depth (1 ft) in the *z* direction (into the page in the figure). In the previous problem, we calculated the initial and final locations of several points on the perimeter of an initially square fluid particle. At $t = 0$, the particle volume is

*Fluid particle volume at t = 0 s*: $$V = \left(0.50\ \text{ft}\right)\left(0.50\ \text{ft}\right)\left(1.0\ \text{ft}\right) = 0.25\ \text{ft}^3 \qquad (1)$$

At $t = 0.2$ s, the lower left corner of the fluid particle has moved to $x = 2.2679$ ft, $y = 0.29889$ ft, and the upper right corner has moved to $x = 3.5225$ ft, $y = 0.49815$ ft. Since the fluid particle remains rectangular, we can calculate the fluid particle volume from these two corner locations,

*Fluid particle volume at t = 0.2 s*:

$$V = \left(3.5225\ \text{ft} - 2.2679\ \text{ft}\right)\left(0.49815\ \text{ft} - 0.29889\ \text{ft}\right)\left(1.0\ \text{ft}\right) = 0.2500\ \text{ft}^3 \qquad (2)$$

Thus, to at least four significant digits, the fluid particle volume has not changed, and **the flow is therefore incompressible**.

***Discussion*** The fluid particle stretches in the horizontal direction and shrinks in the vertical direction, but the net volume of the fluid particle does not change.

**4-51**

**Solution** For a given velocity field we are to use volumetric strain rate to verify that the flow field is incompressible..

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity field is

*Velocity field*:
$$\vec{V} = (u, v) = (U_0 + bx)\vec{i} - by\vec{j} \qquad (1)$$

We use the equation for volumetric strain rate in Cartesian coordinates, and apply Eq. 1,

*Volumetric strain rate*:
$$\frac{1}{V}\frac{DV}{Dt} = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = b + (-b) + 0 = 0 \qquad (2)$$

Where $\varepsilon_{zz}$ = 0 since the flow is two-dimensional. Since the volumetric strain rate is zero everywhere, **the flow is incompressible**.

***Discussion*** The fluid particle stretches in the horizontal direction and shrinks in the vertical direction, but the net volume of the fluid particle does not change.

**4-52**

**Solution** For a given steady two-dimensional velocity field, we are to calculate the *x* and *y* components of the acceleration field.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity field is

$$\vec{V} = (u, v) = (U + a_1 x + b_1 y)\vec{i} + (V + a_2 x + b_2 y)\vec{j} \qquad (1)$$

The acceleration field is obtained from its definition (the material acceleration). The *x*-component is

*x-component of material acceleration*:
$$a_x = \underbrace{\cancel{\frac{\partial u}{\partial t}}}_{\text{Steady}} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + \underbrace{\cancel{w\frac{\partial u}{\partial z}}}_{\text{Two-D}} = (U + a_1 x + b_1 y)a_1 + (V + a_2 x + b_2 y)b_1 \qquad (2)$$

The *y*-component is

*y-component of material acceleration*:
$$a_y = \underbrace{\cancel{\frac{\partial v}{\partial t}}}_{\text{Steady}} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + \underbrace{\cancel{w\frac{\partial v}{\partial z}}}_{\text{Two-D}} = (U + a_1 x + b_1 y)a_2 + (V + a_2 x + b_2 y)b_2 \qquad (3)$$

***Discussion*** If there were a *z*-component, it would be treated in the same fashion.

## 4-53

**Solution** We are to find a relationship among the coefficients that causes the flow field to be incompressible.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** We use the equation for volumetric strain rate in Cartesian coordinates, and apply Eq. 1 of the previous problem,

*Volumetric strain rate*:
$$\frac{1}{V}\frac{DV}{Dt} = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \underbrace{\frac{\partial w}{\partial z}}_{\text{Two-D}} = a_1 + b_2 \qquad (1)$$

We recognize that when the volumetric strain rate is zero everywhere, the flow is incompressible. Thus, the desired relationship is

*Relationship to ensure incompressibility*:
$$\boxed{a_1 + b_2 = 0} \qquad (2)$$

***Discussion*** If Eq. 2 is satisfied, the flow is incompressible, regardless of the values of the other coefficients.

---

## 4-54

**Solution** For a given velocity field we are to calculate the linear strain rates in the *x* and *y* directions.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** We use the equations for linear strain rates in Cartesian coordinates, and apply Eq. 1 of Problem 4-52,

*Linear strain rates*:
$$\boxed{\varepsilon_{xx} = \frac{\partial u}{\partial x} = a_1 \qquad \varepsilon_{yy} = \frac{\partial v}{\partial y} = b_2} \qquad (1)$$

***Discussion*** In general, since coefficients $a_1$ and $b_2$ are non-zero, fluid particles stretch (or shrink) in the *x* and *y* directions.

---

## 4-55

**Solution** For a given velocity field we are to calculate the shear strain rate in the *x*-*y* plane.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** We use the equation for shear strain rate $\varepsilon_{xy}$ in Cartesian coordinates, and apply Eq. 1 of Problem 4-52,

*Shear strain rate in x-y plane*:
$$\boxed{\varepsilon_{xy} = \varepsilon_{yx} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) = \frac{1}{2}\left(b_1 + a_2\right)} \qquad (1)$$

Note that by symmetry $\varepsilon_{yx} = \varepsilon_{xy}$.

***Discussion*** In general, since coefficients $b_1$ and $a_2$ are non-zero, fluid particles distort via shear strain in the *x* and *y* directions.

---

**4-56**

**Solution** For a given velocity field we are to form the 2-D strain rate tensor and determine the conditions necessary for the *x* and *y* axes to be principal axes.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x-y* plane.

***Analysis*** The two-dimensional form of the strain rate tensor is

*2-D strain rate tensor*:

$$\varepsilon_{ij} = \begin{pmatrix} \varepsilon_{xx} & \varepsilon_{xy} \\ \varepsilon_{yx} & \varepsilon_{yy} \end{pmatrix} \qquad (1)$$

We use the linear strain rates and the shear strain rate from the previous two problems to generate the tensor,

*2-D strain rate tensor*:

$$\boxed{\varepsilon_{ij} = \begin{pmatrix} \varepsilon_{xx} & \varepsilon_{xy} \\ \varepsilon_{yx} & \varepsilon_{yy} \end{pmatrix} = \begin{pmatrix} a_1 & \frac{1}{2}\left(b_1 + a_2\right) \\ \frac{1}{2}\left(b_1 + a_2\right) & b_2 \end{pmatrix}} \qquad (2)$$

If the *x* and *y* axes were principal axes, the diagonals of $\varepsilon_{ij}$ would be non-zero, and the off-diagonals would be zero. Here the off-diagonals go to zero when

*Condition for x and y axes to be principal axes*:

$$\boxed{b_1 + a_2 = 0} \qquad (3)$$

***Discussion*** For the more general case in which Eq. 3 is not satisfied, the principal axes can be calculated using tensor algebra.

---

**4-57**

**Solution** For a given velocity field we are to calculate the vorticity vector and discuss its orientation.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x-y* plane.

***Analysis*** We use the equation for vorticity vector $\vec{\zeta}$ in Cartesian coordinates, and apply Eq. 1 of Problem 4-52,

*Vorticity vector*:

$$\boxed{\vec{\zeta} = \left(\underbrace{\frac{\partial w}{\partial y}}_{\text{Two-D}} - \underbrace{\frac{\partial v}{\partial z}}_{\text{Two-D}}\right)\vec{i} + \left(\underbrace{\frac{\partial u}{\partial z}}_{\text{Two-D}} - \underbrace{\frac{\partial w}{\partial x}}_{\text{Two-D}}\right)\vec{j} + \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\vec{k} = \left(a_2 - b_1\right)\vec{k}} \qquad (1)$$

The only non-zero component of vorticity is in the **z (or –z) direction**.

***Discussion*** For any two-dimensional flow in the *x-y* plane, the vorticity vector *must* point in the *z* (or –*z*) direction. The sign of the *z*-component of vorticity in Eq. 1 obviously depends on the sign of $a_2 - b_1$.

---

**4-58**

**Solution** For the given velocity field we are to calculate the two-dimensional linear strain rates from fundamental principles and compare with the given equation.

***Assumptions*** **1** The flow is incompressible. **2** The flow is steady. **3** The flow is two-dimensional.

***Analysis*** First, for convenience, we number the equations in the problem statement:

*Velocity field*: $$\vec{V} = (u, v) = (a + by)\vec{i} + 0\vec{j} \quad (1)$$

*Lower left corner at t + dt*: $$\left(x + (a + by)dt, y\right) \quad (2)$$

*Linear strain rate in Cartesian coordinates*: $$\varepsilon_{xx} = \frac{\partial u}{\partial x} \qquad \varepsilon_{yy} = \frac{\partial v}{\partial y} \quad (3)$$

(*a*) The lower right corner of the fluid particle moves the same amount as the lower left corner since *u* does not depend on *y* position. Thus,

*Lower right corner at t + dt*: $$\left(x + dx + (a + by)dt, y\right) \quad (4)$$

Similarly, the top two corners of the fluid particle move to the right at speed $a + b(y+dy)dt$. Thus,

*Upper left corner at t + dt*: $$\left(x + \left(a + b(y + dy)\right)dt, y + dy\right) \quad (5)$$

and

*Upper right corner at t + dt*: $$\left(x + dx + \left(a + b(y + dy)\right)dt, y + dy\right) \quad (6)$$

(*b*) From the fundamental definition of linear strain rate in the *x*-direction, we consider the lower edge of the fluid particle. Its rate of increase in length divided by its original length is found by using Eqs. 2 and 4,

$\varepsilon_{xx}$: $$\varepsilon_{xx} = \frac{1}{dt}\left[\frac{\overbrace{x + dx + (a + by)dt - \left(x + (a + by)dt\right)}^{\text{Length of lower edge at } t+dt} - \overbrace{dx}^{\text{Length of lower edge at } t}}{dx}\right] = 0 \quad (6)$$

We get the same result by considering the *upper* edge of the fluid particle. Similarly, using the left edge of the fluid particle and Eqs. 2 and 5 we get

$\varepsilon_{yy}$: $$\varepsilon_{yy} = \frac{1}{dt}\left[\frac{\overbrace{y + dy - y}^{\text{Length of left edge at } t+dt} - \overbrace{dy}^{\text{Length of left edge at } t}}{dy}\right] = 0 \quad (7)$$

We get the same result by considering the *right* edge of the fluid particle. Thus both the *x*- and *y*-components of linear strain rate are zero for this flow field.

(*c*) From Eq. 3 we calculate

*Linear strain rates*: $$\boxed{\varepsilon_{xx} = \frac{\partial u}{\partial x} = 0 \qquad \varepsilon_{yy} = \frac{\partial v}{\partial y} = 0} \quad (8)$$

***Discussion*** Although the algebra in this problem is rather straight-forward, it is good practice for the more general case (a later problem).

**4-59**

**Solution** We are to verify that the given flow field is incompressible using two different methods.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional.

***Analysis***

(*a*) The volume of the fluid particle at time *t* is.

*Volume at time t*: $$V(t) = dxdydz \quad (1)$$

where *dz* is the length of the fluid particle in the *z* direction. At time *t* + *dt*, we assume that the fluid particle's dimension *dz* remains fixed since the flow is two-dimensional. Thus its volume is *dz* times the area of the rhombus shown in Fig. P4-58, as illustrated in Fig. 1,

*Volume at time t + dt*: $$V(t+dt) = dxdydz \quad (2)$$

Since Eqs. 1 and 2 are equal, the volume of the fluid particle has not changed, and **the flow is therefore incompressible**.

(*b*) We use the equation for volumetric strain rate in Cartesian coordinates, and apply the results of the previous problem,

*Volumetric strain rate*: $$\frac{1}{V}\frac{DV}{Dt} = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz} = 0+0+0=0 \quad (3)$$

Where $\varepsilon_{zz}$ = 0 since the flow is two-dimensional. Since the volumetric strain rate is zero everywhere, **the flow is incompressible**.

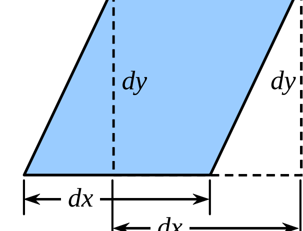



**FIGURE 1**
The area of a rhombus is equal to its base times its height, which here is *dxdy*.

***Discussion*** Although the fluid particle deforms with time, its height, its depth, and the length of its horizontal edges remain constant.

**4-60**

**Solution** For the given velocity field we are to calculate the two-dimensional shear strain rate in the *x*-*y* plane from fundamental principles and compare with the given equation.

***Assumptions*** **1** The flow is incompressible. **2** The flow is steady. **3** The flow is two-dimensional.

***Analysis***

(*a*) The shear strain rate is

*Shear strain rate in Cartesian coordinates*: $$\varepsilon_{xy} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \qquad (1)$$

From the fundamental definition of shear strain rate in the *x*-*y* plane, we consider the bottom edge and the left edge of the fluid particle, which intersect at 90° at the lower left corner at time *t*. We define angle $\alpha$ between the lower edge and the left edge of the fluid particle, and angle $\beta$, the complement of $\alpha$ (Fig. 1). The rate of decrease of angle $\alpha$ over time interval *dt* is obtained from application of trigonometry. First, we calculate angle $\beta$,

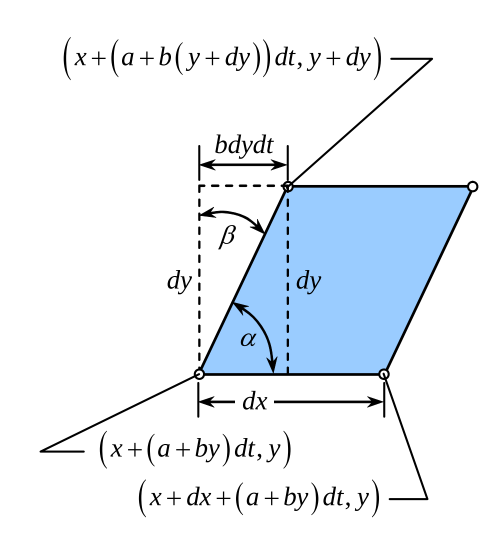



**FIGURE 1**
A magnified view of the deformed fluid particle at time *t* + *dt*, with the location of three corners indicated, and angles $\alpha$ and $\beta$ defined.

*Angle* $\beta$ *at time t + dt*: $$\beta = \arctan\left(\frac{bdydt}{dy}\right) = \arctan(bdt) \approx bdt \qquad (2)$$

The approximation is valid for very small angles. As the time interval $dt \to 0$, Eq. 2 is correct. At time *t* + *dt*, angle $\alpha$ is

*Angle* $\alpha$ *at time t + dt*: $$\alpha = \frac{\pi}{2} - \beta \approx \frac{\pi}{2} - bdt \qquad (3)$$

During this time interval, $\alpha$ changes from 90° ($\pi$/2 radians) to the expression given by Eq. 2. Thus the rate of change of $\alpha$ is

*Rate of change of angle* $\alpha$: $$\frac{d\alpha}{dt} = \frac{1}{dt}\left[\underbrace{\left(\frac{\pi}{2} - bdt\right)}_{\alpha \text{ at } t+dt} - \underbrace{\frac{\pi}{2}}_{\alpha \text{ at } t}\right] = -b \qquad (4)$$

Finally, since shear strain rate is defined as *half* of the rate of *decrease* of angle $\alpha$,

*Shear strain rate*: $$\varepsilon_{xy} = -\frac{1}{2}\frac{d\alpha}{dt} = \boxed{\frac{b}{2}} \qquad (5)$$

(*b*) From Eq. 1 we calculate

*Shear strain rate*: $$\varepsilon_{xy} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) = \frac{1}{2}(b+0) = \boxed{\frac{b}{2}} \qquad (6)$$

Both methods for obtaining the shear strain rate agree (Eq. 5 and Eq. 6).

***Discussion*** Although the algebra in this problem is rather straight-forward, it is good practice for the more general case (a later problem).

**4-61**

**Solution** For the given velocity field we are to calculate the two-dimensional rate of rotation in the *x-y* plane from fundamental principles and compare with the given equation.

***Assumptions*** **1** The flow is incompressible. **2** The flow is steady. **3** The flow is two-dimensional.

***Analysis***

(*a*) The rate of rotation in Cartesian coordinates is

*Rate of rotation in Cartesian coordinates*: $$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) \quad (1)$$

From the fundamental definition of rate of rotation in the *x-y* plane, we consider the bottom edge and the left edge of the fluid particle, which intersect at 90° at the lower left corner at time *t*. We define angle $\beta$ in Fig. 1, where $\beta$ is the negative of the angle of rotation of the left edge of the fluid particle (negative because rotation is mathematically positive in the *counterclockwise* direction). We calculate angle $\beta$ using trigonometry,

*Angle $\beta$ at time t + dt*: $$\beta = \arctan\left(\frac{bdydt}{dy}\right) = \arctan\left(bdt\right) \approx bdt \quad (2)$$

The approximation is valid for very small angles. As the time interval $dt \rightarrow 0$, Eq. 2 is correct. Meanwhile, the bottom edge of the fluid particle has not rotated at all. Thus, the average angle of rotation of the two line segments (lower and left edges) at time *t* + *dt* is

$$AVG = \frac{1}{2}\left(0 - \beta\right) \approx -\frac{b}{2}dt \quad (3)$$

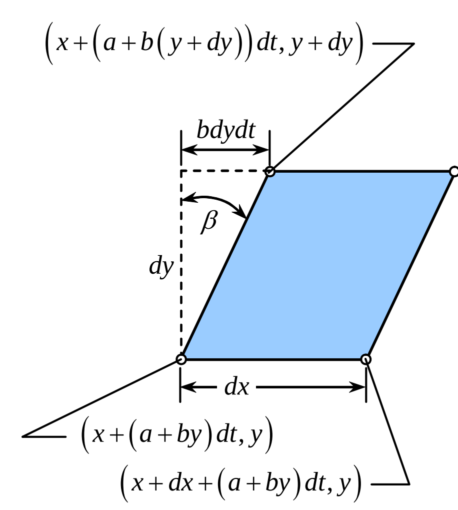



**FIGURE 1**
A magnified view of the deformed fluid particle at time *t* + *dt*, with the location of three corners indicated, and angle $\beta$ defined.

Thus the average rotation *rate* during time interval *dt* is

*Rate of rotation in x-y plane*: $$\omega_z = \frac{d\left(AVG\right)}{dt} = \frac{1}{dt}\left(-\frac{b}{2}dt\right) = \boxed{-\frac{b}{2}} \quad (4)$$

(*b*) From Eq. 1 we calculate

*Rate of rotation*: $$\omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) = \frac{1}{2}\left(0 - b\right) = \boxed{-\frac{b}{2}} \quad (5)$$

Both methods for obtaining the rate of rotation agree (Eq. 4 and Eq. 5).

***Discussion*** The rotation rate is negative, indicating *clockwise* rotation about the *z*-axis. This agrees with our intuition as we follow the fluid particle.

---

**4-62**

**Solution** We are to determine whether the shear flow of Problem 4-22 is rotational or irrotational, and we are to calculate the vorticity in the *z* direction.

***Analysis***

(*a*) Since the rate of rotation is non-zero, it means that **the flow is *rotational***.

(*b*) Vorticity is defined as twice the rate of rotation, or twice the angular velocity. In the *z* direction,

*Vorticity component*: $$\boxed{\zeta_z = 2\omega_z = 2\left(-\frac{b}{2}\right) = -b} \quad (1)$$

***Discussion*** Vorticity is negative, indicating *clockwise* rotation about the *z*-axis.

---

**4-63**

**Solution** We are to prove the given expression for flow in the *xy*-plane.

***Assumptions*** **1** The flow is incompressible and two-dimensional.

***Analysis*** For flow in the *xy*-plane, we are to show that:

*Rate of rotation*:
$$\omega = \omega_z = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) \qquad (1)$$

By definition, the rate of rotation (angular velocity) at a point is the average rotation rate of two initially perpendicular lines that intersect at the point. In this particular problem, Line a (PA) and Line b (PB) are initially perpendicular, and intersect at point P. Line a rotates by angle $\alpha_a$, and Line b rotates by angle $\alpha_b$, Thus, the average angle of rotation is

*Average angle of rotation*:
$$\frac{\alpha_a + \alpha_b}{2} \qquad (2)$$

During time increment *dt*, point P moves a distance *udt* to the right and *vdt* up (to first order, assuming *dt* is very small). Similarly, point A moves a distance $\left(u + \frac{\partial u}{\partial x}dx\right)dt$ to the right and $\left(v + \frac{\partial v}{\partial x}dx\right)dt$ up, and point B moves a distance $\left(u + \frac{\partial u}{\partial y}dy\right)dt$ to the right and $\left(v + \frac{\partial v}{\partial y}dy\right)dt$ up. Since point A is initially at distance *dx* to the right of point P, the horizontal distance from point P′ to point A′ at the later time $t_2$ is

$$dx + \frac{\partial u}{\partial x}dxdt$$

On the other hand, point A is at the same *vertical* level as point P at time $t_1$. Thus, the vertical distance from point P′ to point A′ at time $t_2$ is

$$\frac{\partial v}{\partial x}dxdt \qquad (3)$$

Similarly, point B is located at distance *dy* vertically above point P at time $t_1$, and thus the horizontal distance from point P′ to point B′ at time $t_2$ is

$$-\frac{\partial u}{\partial y}dydt \qquad (4)$$

and

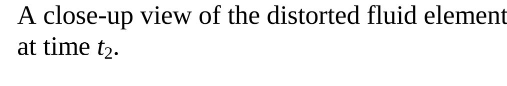



**FIGURE 1**
A close-up view of the distorted fluid element at time $t_2$.

*Vertical distance from point P′ to point B′ at time* $t_2$:
$$dy + \frac{\partial v}{\partial y}dydt \qquad (5)$$

We mark the horizontal and vertical distances between point A′ and point P′ and between point B′ and point P′ at time $t_2$ in Fig. 1. From the figure we see that

*Angle* $\alpha_a$ *in terms of velocity components*:
$$\alpha_a = \tan^{-1}\left(\frac{\frac{\partial v}{\partial x}dxdt}{dx + \frac{\partial u}{\partial x}dxdt}\right) \approx \tan^{-1}\left(\frac{\frac{\partial v}{\partial x}dxdt}{dx}\right) = \tan^{-1}\left(\frac{\partial v}{\partial x}dt\right) \approx \frac{\partial v}{\partial x}dt \qquad (6)$$

The first approximation in Eq. 6 is due to the fact that as the size of the fluid element shrinks to a point, $dx \to 0$, and at the same time $dt \to 0$. Thus, the second term in the denominator is second-order compared to the first-order term *dx* and can be neglected. The second approximation in Eq. 6 is because as $dt \to 0$ angle $\alpha_a$ is very small, and $\tan\alpha_a \to \alpha_a$. Similarly, *angle* $\alpha_b$ is written in terms of velocity components as

$$\alpha_\mathrm{b} = \tan^{-1}\left(\frac{-\frac{\partial u}{\partial y}dydt}{dy+\frac{\partial v}{\partial y}dydt}\right) \approx \tan^{-1}\left(\frac{-\frac{\partial u}{\partial y}dydt}{dy}\right) = \tan^{-1}\left(-\frac{\partial u}{\partial y}dt\right) \approx -\frac{\partial u}{\partial y}dt \qquad (7)$$

Finally then, the average rotation angle (Eq. 2) becomes

*Average angle of rotation*:

$$\frac{\alpha_\mathrm{a}+\alpha_\mathrm{b}}{2} = \frac{1}{2}\left(\frac{\partial v}{\partial x}dt - \frac{\partial u}{\partial y}dt\right) = \frac{dt}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) \qquad (8)$$

and the average *rate* of rotation (angular velocity) of the fluid element about point P in the *x*-*y* plane becomes

$$\boxed{\omega = \omega_z = \frac{d}{dt}\left(\frac{\alpha_\mathrm{a}+\alpha_\mathrm{b}}{2}\right) = \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)} \qquad (9)$$

***Discussion*** Eq. 9 can be extended to three dimensions by performing a similar analysis in the *x*-*z* and *y*-*z* planes.

## 4-64

**Solution** We are to prove the given expression.

***Assumptions*** **1** The flow is incompressible and two-dimensional.

***Analysis*** We are to prove the following:

*Linear strain rate in x-direction*: $$\varepsilon_{xx} = \frac{\partial u}{\partial x} \tag{1}$$

By definition, the rate of linear strain is the rate of increase in length of a line segment in a given direction divided by the original length of the line segment in that direction. During time increment $dt$, point P moves a distance $udt$ to the right and $vdt$ up (to first order, assuming $dt$ is very small). Similarly, point A moves a distance $\left(u + \frac{\partial u}{\partial x}dx\right)dt$ to the right and $\left(v + \frac{\partial v}{\partial x}dx\right)dt$ up. Since point A is initially at distance $dx$ to the right of point P, its position to the right of point P′ at the later time $t_2$ is

$$dx + \frac{\partial u}{\partial x}dxdt \tag{2}$$

On the other hand, point A is at the same *vertical* level as point P at time $t_1$. Thus, the vertical distance from point P′ to point A′ at time $t_2$ is

*Vertical distance from point P′ to point A′ at time $t_2$*: $$\frac{\partial v}{\partial x}dxdt \tag{3}$$

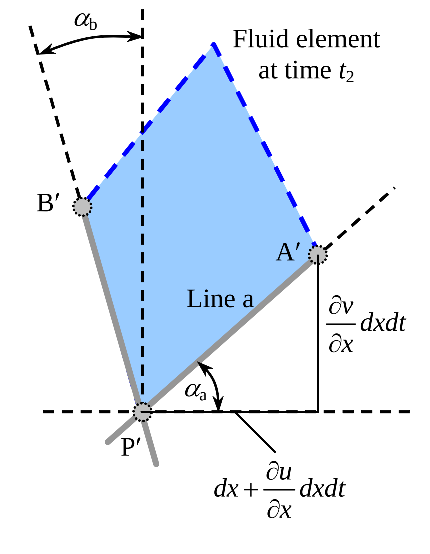



**FIGURE 1**
A close-up view of the distorted fluid element at time $t_2$.

We mark the horizontal and vertical distances between point A′ and point P′ at time $t_2$ in Fig. 1. From the figure we see that

*Linear strain rate in the x direction as line PA changes to P′A′*:

$$\varepsilon_{xx} = \frac{d}{dt}\left( \frac{\overbrace{dx + \frac{\partial u}{\partial x}dxdt}^{\text{Length of P′A′ in } x \text{ direction}} - \overbrace{dx}^{\text{Length of PA in } x \text{ direction}}}{\underbrace{dx}_{\text{Length of PA in } x \text{ direction}}} \right) = \frac{d}{dt}\left(\frac{\partial u}{\partial x}dt\right) = \frac{\partial u}{\partial x} \tag{4}$$

Thus **Eq. 1 is verified**.

***Discussion*** The distortion of the fluid element is exaggerated in Fig. 1. As time increment $dt$ and fluid element length $dx$ approach zero, the first-order approximations become exact.

## 4-65

**Solution** We are to prove the given expression.

***Assumptions*** **1** The flow is incompressible and two-dimensional.

***Analysis*** We are to prove the following:

*Shear strain rate in xy-plane*: $$\varepsilon_{xy} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \qquad (1)$$

By definition, the shear strain rate at a point is half of the rate of decrease of the angle between two initially perpendicular lines that intersect at the point. In Fig. P4-63, Line a (PA) and Line b (PB) are initially perpendicular, and intersect at point P. Line a rotates by angle $\alpha_a$, and Line b rotates by angle $\alpha_b$. The angle between these two lines changes from $\pi/2$ at time $t_1$ to $\alpha_{a\text{-}b}$ at time $t_2$ as sketched in Fig. 1. The shear strain rate at point P for initially perpendicular lines in the $x$ and $y$ directions is thus

$$\varepsilon_{xy} = -\frac{1}{2}\frac{d}{dt}\alpha_{a\text{-}b} \qquad (2)$$

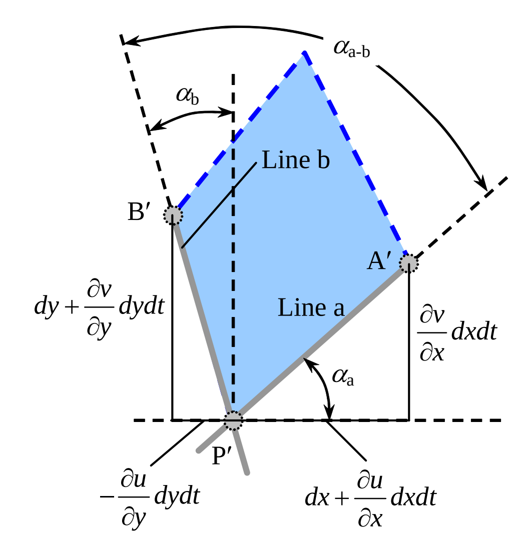



**FIGURE 1**
A close-up view of the distorted fluid element at time $t_2$.

During time increment $dt$, point P moves a distance $udt$ to the right and $vdt$ up (to first order, assuming $dt$ is very small). Similarly, point A moves a distance $\left(u + \frac{\partial u}{\partial x}dx\right)dt$ to the right and $\left(v + \frac{\partial v}{\partial x}dx\right)dt$ up, and point B moves a distance $\left(u + \frac{\partial u}{\partial y}dy\right)dt$ to the right and $\left(v + \frac{\partial v}{\partial y}dy\right)dt$ up. Since point A is initially at distance $dx$ to the right of point P, its position to the right of point P′ at the later time $t_2$ is

*Horizontal distance from point P′ to point A′ at time* $t_2$: $$dx + \frac{\partial u}{\partial x}dxdt \qquad (3)$$

On the other hand, point A is at the same *vertical* level as point P at time $t_1$. Thus, the vertical distance from point P′ to point A′ at time $t_2$ is

*Vertical distance from point P′ to point A′ at time* $t_2$: $$\frac{\partial v}{\partial x}dxdt \qquad (3)$$

Similarly, point B is located at distance $dy$ vertically above point P at time $t_1$, and thus we write

*Horizontal distance from point P′ to point B′ at time* $t_2$: $$-\frac{\partial u}{\partial y}dydt \qquad (4)$$

and

*Vertical distance from point P′ to point B′ at time* $t_2$: $$dy + \frac{\partial v}{\partial y}dydt \qquad (5)$$

We mark the horizontal and vertical distances between point A′ and point P′ and between point B′ and point P′ at time $t_2$ in Fig. 1. From the figure we see that

*Angle $\alpha_a$ in terms of velocity components*:
$$\alpha_a = \tan^{-1}\left(\frac{\frac{\partial v}{\partial x}dxdt}{dx + \frac{\partial u}{\partial x}dxdt}\right) \approx \tan^{-1}\left(\frac{\frac{\partial v}{\partial x}dxdt}{dx}\right) = \tan^{-1}\left(\frac{\partial v}{\partial x}dt\right) \approx \frac{\partial v}{\partial x}dt \qquad (6)$$

The first approximation in Eq. 6 is due to the fact that as the size of the fluid element shrinks to a point, $dx \rightarrow 0$, and at the same time $dt \rightarrow 0$. Thus, the second term in the denominator is second-order compared to the first-order term $dx$ and can be neglected. The second approximation in Eq. 6 is because as $dt \rightarrow 0$ angle $\alpha_a$ is very small, and $\tan\alpha_a \rightarrow \alpha_a$. Similarly,

*Angle $\alpha_b$ in terms of velocity components*:

$$\alpha_b = \tan^{-1}\left(\frac{-\frac{\partial u}{\partial y}dydt}{dy+\frac{\partial v}{\partial y}dydt}\right) \approx \tan^{-1}\left(\frac{-\frac{\partial u}{\partial y}dydt}{dy}\right) = \tan^{-1}\left(-\frac{\partial u}{\partial y}dt\right) \approx -\frac{\partial u}{\partial y}dt \qquad (7)$$

Angle $\alpha_{a\text{-}b}$ at time $t_2$ is calculated from Fig. 1 as

*Angle $\alpha_{a\text{-}b}$ at time $t_2$ in terms of velocity components*:

$$\alpha_{a\text{-}b} = \frac{\pi}{2} + \alpha_b - \alpha_a = \frac{\pi}{2} - \frac{\partial u}{\partial y}dt - \frac{\partial v}{\partial x}dt \qquad (8)$$

where we have used Eqs. 6 and 7. Finally then, the shear strain rate (Eq. 2) becomes

*Shear strain rate, initially perpendicular lines in the x and y directions*:

$$\varepsilon_{xy} = -\frac{1}{2}\frac{d}{dt}\alpha_{a\text{-}b} \approx -\frac{1}{2}\frac{1}{dt}\left(\overbrace{\frac{\pi}{2} - \frac{\partial u}{\partial y}dt - \frac{\partial v}{\partial x}dt}^{\alpha_{a\text{-}b} \text{ at } t_2} - \overbrace{\frac{\pi}{2}}^{\alpha_{a\text{-}b} \text{ at } t_1}\right) = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \qquad (9)$$

which agrees with Eq. 1. Thus, **Eq. 1 is proven**.

***Discussion*** Eq. 9 can be easily extended to three dimensions by performing a similar analysis in the *x*-*z* plane and in the *y*-*z* plane.

---

## 4-66

**Solution** For a given linear strain rate in the *x*-direction, we are to calculate the linear strain rate in the *y*-direction.

***Analysis*** Since the flow is incompressible, the volumetric strain rate must be zero. In two dimensions,

*Volumetric strain rate in the x-y plane*:

$$\frac{1}{V}\frac{DV}{Dt} = \varepsilon_{xx} + \varepsilon_{yy} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad (1)$$

Thus, the linear strain rate in the *y*-direction is the negative of that in the *x*-direction,

*Linear strain rate in y-direction*:

$$\varepsilon_{yy} = \frac{\partial v}{\partial y} = -\frac{\partial u}{\partial x} = \mathbf{-2.5\ 1/s} \qquad (2)$$

***Discussion*** The fluid element *stretches* in the *x*-direction since $\varepsilon_{xx}$ is positive. Because the flow is incompressible, the fluid element must *shrink* in the *y*-direction, yielding a value of $\varepsilon_{yy}$ that is negative.

---

**4-67**

**Solution** We are to calculate the vorticity of fluid particles in a tank rotating in solid body rotation about its vertical axis.

***Assumptions*** **1** The flow is steady. **2** The *z*-axis is in the vertical direction.

***Analysis*** Vorticity $\vec{\zeta}$ is twice the angular velocity $\vec{\omega}$. Here,

*Angular velocity*:
$$\vec{\omega} = 360\frac{\text{rot}}{\text{min}}\left(\frac{1\text{ min}}{60\text{ s}}\right)\left(\frac{2\pi\text{ rad}}{\text{rot}}\right)\vec{k} = 37.70\vec{k}\text{ rad/s} \qquad (1)$$

where $\vec{k}$ is the unit vector in the vertical ($z$) direction. The vorticity is thus

*Vorticity*:
$$\boxed{\vec{\zeta} = 2\vec{\omega} = 2\times 37.70\vec{k}\text{ rad/s} = 75.4\vec{k}\text{ rad/s}} \qquad (2)$$

***Discussion*** Because the water rotates as a solid body, the vorticity is constant throughout the tank, and points vertically upward.

---

**4-68**

**Solution** We are to calculate the angular speed of a tank rotating about its vertical axis.

***Assumptions*** **1** The flow is steady. **2** The *z*-axis is in the vertical direction.

***Analysis*** Vorticity $\vec{\zeta}$ is twice the angular velocity $\vec{\omega}$. Thus,

*Angular velocity*:
$$\vec{\omega} = \frac{\vec{\zeta}}{2} = \frac{-55.4\vec{k}\text{ rad/s}}{2} = -27.7\vec{k}\text{ rad/s} \qquad (1)$$

where $\vec{k}$ is the unit vector in the vertical ($z$) direction. The angular velocity is negative, which by definition is in the clockwise direction about the vertical axis. We express the rate of rotation in units of rpm,

*Rate of rotation*:
$$\dot{n} = -27.7\frac{\text{rad}}{\text{s}}\left(\frac{60\text{ s}}{1\text{ min}}\right)\left(\frac{\text{rot}}{2\pi\text{ rad}}\right) = -265\frac{\text{rot}}{\text{min}} = \mathbf{-265\text{ rpm}} \qquad (2)$$

***Discussion*** Because the vorticity is constant throughout the tank, the water rotates as a solid body.

---

**4-69**

**Solution** For a tank of given rim radius and speed, we are to calculate the magnitude of the component of vorticity in the vertical direction.

***Assumptions*** **1** The flow is steady. **2** The *z*-axis is in the vertical direction.

***Analysis*** The linear speed at the rim is equal to $r_{\text{rim}}\omega_z$. Thus,

*Component of angular velocity in z-direction*:
$$\omega_z = \frac{V_{\text{rim}}}{r_{\text{rim}}} = \frac{2.6\text{ m/s}}{0.35\text{ m}} = 7.429\text{ rad/s} \qquad (1)$$

Vorticity $\vec{\zeta}$ is twice the angular velocity $\vec{\omega}$. Thus,

*z-component of vorticity*:
$$\zeta_z = 2\omega_z = 2(7.429\text{ rad/s}) = 14.86\text{ rad/s} \cong \mathbf{15.0\text{ rad/s}} \qquad (2)$$

***Discussion*** Radian is a non-dimensional unit, so we can insert it into Eq. 1. The final answer is given to two significant digits for consistency with the given information.

---

## 4-70C

**Solution** We are to explain the relationship between vorticity and rotationality.

***Analysis*** Vorticity is a measure of the rotationality of a fluid particle. If a particle rotates, its vorticity is non-zero. Mathematically, **the vorticity vector is twice the angular velocity vector**.

***Discussion*** If the vorticity is zero, the flow is called irrotational.

---

## 4-71

**Solution** For a given deformation of a fluid particle in one direction, we are to calculate its deformation in the other direction.

***Assumptions*** **1** The flow is incompressible. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** Since the flow is incompressible and two-dimensional, the area of the fluid element must remain constant (volumetric strain rate must be zero in an incompressible flow). The area of the original fluid particle is $a^2$. Hence, **the vertical dimension of the fluid particle at the later time must be $a^2/2a = a/2$**.

***Discussion*** Since the particle stretches by a factor of two in the *x*-direction, it shrinks by a factor of two in the *y*-direction.

---

## 4-72

**Solution** We are to calculate the percentage change in fluid density for a fluid particle undergoing two-dimensional deformation.

***Assumptions*** **1** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The area of the original fluid particle is $a^2$. Assuming that the mass of the fluid particle is *m* and its dimension in the *z*-direction is also *a*, the initial density is $\rho = m/V = m/a^3$. As the particle moves and deforms, its mass must remain constant. If its dimension in the *z*-direction remains equal to *a*, the density at the later time is

*Density at the later time*:
$$\rho = \frac{m}{V} = \frac{m}{(1.06a)(0.931a)(a)} = 1.013\frac{m}{a^3} \qquad (1)$$

Compared to the original density, **the density has increased by about 1.3%**.

***Discussion*** The fluid particle has stretched in the *x*-direction and shrunk in the *y*-direction, but there is nevertheless a net decrease in volume, corresponding to a net increase in density.

---

## 4-73

**Solution** For a given velocity field we are to calculate the vorticity.

***Analysis*** The velocity field is

$$\vec{V} = (u, v, w) = (3.0 + 2.0x - y)\vec{i} + (2.0x - 2.0y)\vec{j} + (0.5xy)\vec{k} \qquad (1)$$

In Cartesian coordinates, the vorticity vector is

*Vorticity vector in Cartesian coordinates*:
$$\vec{\zeta} = \left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right)\vec{i} + \left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right)\vec{j} + \left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right)\vec{k} \qquad (2)$$

We substitute the velocity components $u = 3.0 + 2.0x - y$, $v = 2.0x - 2.0y$, and $w = 0.5xy$ from Eq. 1 into Eq. 2 to obtain

*Vorticity vector*:
$$\boxed{\vec{\zeta} = (0.5x - 0)\vec{i} + (0 - 0.5y)\vec{j} + (2.0 - (-1))\vec{k} = (0.5x)\vec{i} - (0.5y)\vec{j} + (3.0)\vec{k}} \qquad (3)$$

***Discussion*** The vorticity is non-zero implying that this flow field is *rotational*.

---

## 4-74

**Solution** We are to determine if the flow is rotational, and if so calculate the *z*-component of vorticity.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity field is given by

*Velocity field, Couette flow*:
$$\vec{V} = (u, v) = \left(V\frac{y}{h}\right)\vec{i} + 0\vec{j} \qquad (1)$$

If the vorticity is non-zero, the flow is rotational. So, we calculate the *z*-component of vorticity,

*z-component of vorticity*:
$$\boxed{\zeta_z = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 - \frac{V}{h} = -\frac{V}{h}} \qquad (2)$$

Since vorticity is non-zero, **this flow is rotational**. Furthermore, the vorticity is negative, implying that **particles rotate in the clockwise direction**.

***Discussion*** The vorticity is constant at every location in this flow.

---

## 4-75

**Solution** For the given velocity field for Couette flow, we are to calculate the two-dimensional linear strain rates and the shear strain rate.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The linear strain rates in the *x* direction and in the *y* direction are

*Linear strain rates*:
$$\varepsilon_{xx} = \frac{\partial u}{\partial x} = \mathbf{0} \qquad \varepsilon_{yy} = \frac{\partial v}{\partial y} = \mathbf{0} \qquad (1)$$

The shear strain rate in the *x*-*y* plane is

*Shear strain rate*:
$$\boxed{\varepsilon_{xy} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) = \frac{1}{2}\left(\frac{V}{h} + 0\right) = \frac{V}{2h}} \qquad (2)$$

Fluid particles in this flow have non-zero shear strain rate.

***Discussion*** Since the linear strain rates are zero, fluid particles deform (shear), but do not *stretch* in either the horizontal or vertical directions.

---

**4-76**

**Solution** For the Couette flow velocity field we are to form the 2-D strain rate tensor and determine if the *x* and *y* axes are principal axes.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The two-dimensional strain rate tensor, $\varepsilon_{ij}$, is

*2-D strain rate tensor*:
$$\varepsilon_{ij} = \begin{pmatrix} \varepsilon_{xx} & \varepsilon_{xy} \\ \varepsilon_{yx} & \varepsilon_{yy} \end{pmatrix} \tag{1}$$

We use the linear strain rates and the shear strain rate from the previous problem to generate the tensor,

*2-D strain rate tensor*:
$$\boxed{\varepsilon_{ij} = \begin{pmatrix} \varepsilon_{xx} & \varepsilon_{xy} \\ \varepsilon_{yx} & \varepsilon_{yy} \end{pmatrix} = \begin{pmatrix} 0 & \dfrac{V}{2h} \\ \dfrac{V}{2h} & 0 \end{pmatrix}} \tag{2}$$

Note that by symmetry $\varepsilon_{yx} = \varepsilon_{xy}$. If the *x* and *y* axes were principal axes, the diagonals of $\varepsilon_{ij}$ would be non-zero, and the off-diagonals would be zero. Here we have the opposite case, so **the x and y axes are not principal axes**.

***Discussion*** The principal axes can be calculated using tensor algebra.

## Reynolds Transport Theorem

### 4-77C

**Solution**

(*a*) **False**: The statement is backwards, since the conservation laws are naturally occurring in the system form.
(*b*) **False**: The RTT can be applied to any control volume, fixed, moving, or deforming.
(*c*) **True**: The RTT has an unsteady term and can be applied to unsteady problems.
(d) **True**: The extensive property *B* (or its intensive form *b*) in the RTT can be any property of the fluid – scalar, vector, or even tensor.

### 4-78

**Solution** For the case in which $B_{sys}$ is the mass *m* of a system, we are to use the RTT to derive the equation of conservation of mass for a control volume.

***Analysis*** The general form of the Reynolds transport theorem is given by

*General form of the RTT*:
$$\frac{dB_{sys}}{dt} = \frac{d}{dt}\int_{CV} \rho b dV + \int_{CS} \rho b \vec{V}_r \cdot \vec{n} dA \qquad (1)$$

Setting $B_{sys} = m$ means that $b = m/m = 1$. Plugging these and $dm/dt = 0$ into Eq. 1 yields

*Conservation of mass for a CV*:
$$0 = \frac{d}{dt}\int_{CV} \rho dV + \int_{CS} \rho \vec{V}_r \cdot \vec{n} dA \qquad (2)$$

***Discussion*** Eq. 2 is general and applies to any control volume – fixed, moving, or even deforming.

### 4-79

**Solution** For the case in which $B_{sys}$ is the linear momentum $m\vec{V}$ of a system, we are to use the RTT to derive the equation of conservation of linear momentum for a control volume.

***Analysis*** Newton's second law is

*Newton's second law for a system*:
$$\sum \vec{F} = m\vec{a} = m\frac{d\vec{V}}{dt} = \frac{d}{dt}\left(m\vec{V}\right)_{sys} \qquad (1)$$

Setting $B_{sys} = m\vec{V}$ means that $b = m\vec{V}/m = \vec{V}$. Plugging these and Eq. 1 into the equation of the previous problem yields

$$\sum \vec{F} = \frac{d}{dt}\left(m\vec{V}\right)_{sys} = \frac{d}{dt}\int_{CV} \rho \vec{V} dV + \int_{CS} \rho \vec{V}\left(\vec{V}_r \cdot \vec{n}\right) dA$$

or simply

*Conservation of linear momentum for a CV*:
$$\sum \vec{F} = \frac{d}{dt}\int_{CV} \rho \vec{V} dV + \int_{CS} \rho \vec{V}\left(\vec{V}_r \cdot \vec{n}\right) dA \qquad (2)$$

***Discussion*** Eq. 2 is general and applies to any control volume – fixed, moving, or even deforming.

## 4-80

**Solution** For the case in which $B_{sys}$ is the angular momentum $\vec{H}$ of a system, we are to use the RTT to derive the equation of conservation of angular momentum for a control volume.

***Analysis*** The conservation of angular momentum is expressed as

*Conservation of angular momentum for a system*: $$\sum \vec{M} = \frac{d}{dt}\vec{H}_{sys} \qquad (1)$$

Setting $B_{sys}$ = $\vec{H}$ means that $b = \left(\vec{r} \times m\vec{V}\right)/m = \vec{r} \times \vec{V}$, noting that $m$ = constant for a system. Plugging these and Eq. 1 into the equation of Problem 4-78 yields

$$\sum \vec{M} = \frac{d}{dt}\vec{H}_{sys} = \frac{d}{dt}\int_{CV} \rho\left(\vec{r} \times \vec{V}\right) dV + \int_{CS} \rho\left(\vec{r} \times \vec{V}\right)\left(\vec{V}_r \cdot \vec{n}\right) dA$$

or simply

*Conservation of angular momentum for a CV*:

$$\boxed{\sum \vec{M} = \frac{d}{dt}\int_{CV} \rho\left(\vec{r} \times \vec{V}\right) dV + \int_{CS} \rho\left(\vec{r} \times \vec{V}\right)\left(\vec{V}_r \cdot \vec{n}\right) dA} \qquad (2)$$

***Discussion*** Eq. 2 is general and applies to any control volume – fixed, moving, or even deforming.

---

## 4-81

**Solution** $F(t)$ is to be evaluated from the given expression.

***Analysis*** The integral is

$$F(t) = \frac{d}{dt}\int_{x=At}^{x=Bt} e^{-2x^2} dx \qquad (1)$$

We could try integrating first, and then differentiating, but we can instead use the 1-D Leibnitz theorem. Here, $G(x,t) = e^{-2x^2}$ ($G$ is not a function of time in this simple example). The limits of integration are $a(t) = At$ and $b(t) = Bt$. Thus,

$$F(t) = \int_a^b \frac{\partial G}{\partial t} dx + \frac{db}{dt}G(b,t) - \frac{da}{dt}G(a,t) \qquad (2)$$
$$= \quad 0 \quad + Be^{-2b^2} \quad - Ae^{-2a^2}$$

or

$$\boxed{F(t) = Be^{-B^2t^2} - Ae^{-A^2t^2}} \qquad (3)$$

***Discussion*** You are welcome to try to obtain the same solution without using the Leibnitz theorem.

---

## Review Problems

### 4-82

**Solution** We are to determine if the flow is rotational, and if so calculate the *z*-component of vorticity.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity components are given by

*Velocity components, 2-D Poiseuille flow*: $$u = \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right) \qquad v = 0 \qquad (1)$$

If the vorticity is non-zero, the flow is rotational. So, we calculate the *z*-component of vorticity,

*z-component of vorticity*:
$$\boxed{\zeta_z = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 - \frac{1}{2\mu}\frac{dP}{dx}\left(2y - h\right) = -\frac{1}{2\mu}\frac{dP}{dx}\left(2y - h\right)} \qquad (2)$$

Since vorticity is non-zero, **this flow is rotational**. Furthermore, in the lower half of the flow ($y < h/2$) the vorticity is negative (note that $dP/dx$ is negative). Thus, **particles rotate in the clockwise direction in the lower half of the flow**. Similarly, **particles rotate in the counterclockwise direction in the upper half of the flow**.

***Discussion*** The vorticity varies linearly across the channel.

### 4-83

**Solution** For the given velocity field for 2-D Poiseuille flow, we are to calculate the two-dimensional linear strain rates and the shear strain rate.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The linear strain rates in the *x* direction and in the *y* direction are

*Linear strain rates*: $$\varepsilon_{xx} = \frac{\partial u}{\partial x} = \mathbf{0} \qquad \varepsilon_{yy} = \frac{\partial v}{\partial y} = \mathbf{0} \qquad (1)$$

The shear strain rate in the *x*-*y* plane is

*Shear strain rate*:
$$\boxed{\varepsilon_{xy} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) = \frac{1}{2}\left(\frac{1}{2\mu}\frac{dP}{dx}\left(2y - h\right) + 0\right) = \frac{1}{4\mu}\frac{dP}{dx}\left(2y - h\right)} \qquad (2)$$

Fluid particles in this flow have non-zero shear strain rate.

***Discussion*** Since the linear strain rates are zero, fluid particles deform (shear), but do not *stretch* in either the horizontal or vertical directions.

**4-84**

**Solution** For the 2-D Poiseuille flow velocity field we are to form the 2-D strain rate tensor and determine if the $x$ and $y$ axes are principal axes.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the $x$-$y$ plane.

***Analysis*** The two-dimensional strain rate tensor, $\varepsilon_{ij}$, in the $x$-$y$ plane,

*2-D strain rate tensor*:

$$\varepsilon_{ij} = \begin{pmatrix} \varepsilon_{xx} & \varepsilon_{xy} \\ \varepsilon_{yx} & \varepsilon_{yy} \end{pmatrix} \qquad (1)$$

We use the linear strain rates and the shear strain rate from the previous problem to generate the tensor,

$$\varepsilon_{ij} = \begin{pmatrix} \varepsilon_{xx} & \varepsilon_{xy} \\ \varepsilon_{yx} & \varepsilon_{yy} \end{pmatrix} = \begin{pmatrix} 0 & \frac{1}{4\mu}\frac{dP}{dx}(2y-h) \\ \frac{1}{4\mu}\frac{dP}{dx}(2y-h) & 0 \end{pmatrix} \qquad (2)$$

Note that by symmetry $\varepsilon_{yx} = \varepsilon_{xy}$. If the $x$ and $y$ axes were principal axes, the diagonals of $\varepsilon_{ij}$ would be non-zero, and the off-diagonals would be zero. Here we have the opposite case, so **the x and y axes are not principal axes**.

***Discussion*** The principal axes can be calculated using tensor algebra.

**4-85** 

**Solution** For a given velocity field we are to plot several pathlines for fluid particles released from various locations and over a specified time period.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the $x$-$y$ plane.

***Properties*** For water at 40°C, $\mu = 6.53 \times 10^{-4}$ kg/m·s.

***Analysis*** Since the flow is steady, pathlines, streamlines, and streaklines are all straight horizontal lines. We simply need to integrate velocity component $u$ with respect to time over the specified time period. The horizontal velocity component is

$$u = \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right) \qquad (1)$$

We integrate as follows:

$$x = x_{\text{start}} + \int_{t_{\text{start}}}^{t_{\text{end}}} u\,dt = 0 + \int_0^{10\text{ s}} \left(\frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right)\right)dt$$
$$x = \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right)(10\text{ s}) \qquad (2)$$

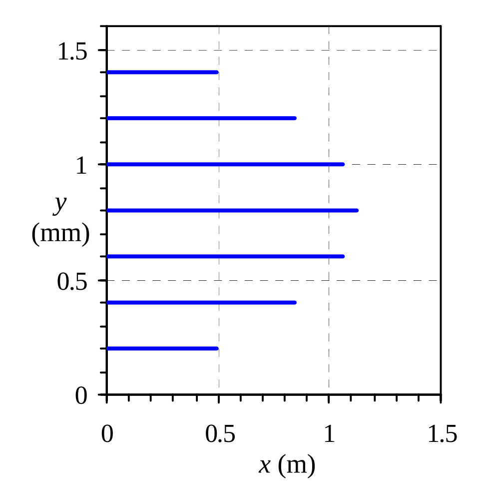



**FIGURE 1**
Pathlines for the given velocity field at $t$ = 12 s. Note that the vertical scale is greatly expanded for clarity ($x$ is in m, but $y$ is in mm).

We substitute the given values of $y$ and the values of $\mu$ and $dP/dx$ into Eq. 2 to calculate the ending $x$ position of each pathline. We plot the pathlines in **Fig. 1**.

***Discussion*** Streaklines introduced at the same locations and developed over the same time period would look identical to the pathlines of Fig. 1.

**4-86** [*Also solved using EES on enclosed DVD*]

**Solution** For a given velocity field we are to plot several streaklines at a given time for dye released from various locations over a specified time period.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Properties*** For water at 40°C, $\mu = 6.53\times10^{-4}$ kg/m·s.

***Analysis*** Since the flow is steady, pathlines, streamlines, and streaklines are all straight horizontal lines. We simply need to integrate velocity component *u* with respect to time over the specified time period. The horizontal velocity component is

$$u = \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right) \qquad (1)$$

We integrate as follows to obtain the final *x* location of the first dye particle released:

$$x = x_{\text{start}} + \int_{t_{\text{start}}}^{t_{\text{end}}} u\,dt = 0 + \int_0^{10\text{ s}}\left(\frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right)\right)dt$$
$$x = \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right)\times\left(10\text{ s}\right) \qquad (2)$$

We substitute the given values of *y* and the values of $\mu$ and $dP/dx$ into Eq. 2 to calculate the ending *x* position of the first released dye particle of each streakline. The *last* released dye particle is at $x = x_{\text{start}} = 0$, because it hasn't had a chance to go anywhere. We connect the beginning and ending points to plot the streaklines (**Fig. 1**).

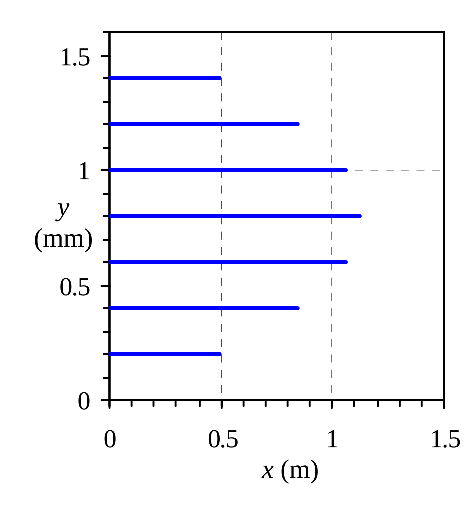



**FIGURE 1**
Streaklines for the given velocity field at *t* = 10 s. Note that the vertical scale is greatly expanded for clarity (*x* is in m, but *y* is in mm).

***Discussion*** These streaklines are introduced at the same locations and are developed over the same time period as the pathlines of the previous problem. They are identical since the flow is steady.

**4-87** 

**Solution** For a given velocity field we are to plot several streaklines at a given time for dye released from various locations over a specified time period.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Properties*** For water at 40°C, $\mu = 6.53 \times 10^{-4}$ kg/m·s.

***Analysis*** Since the flow is steady, pathlines, streamlines, and streaklines are all straight horizontal lines. The horizontal velocity component is

$$u = \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right) \qquad (1)$$

In the previous problem we generated streaklines at $t$ = 10 s. Imagine the dye at the source being suddenly cut off at that time, but the streaklines are observed 2 seconds later, at $t$ = 12 s. The dye streaks will not stretch any further, but will simply move at the same horizontal speed for 2 more seconds. At each $y$ location, the $x$ locations of the first and last dye particle are thus

*first dye particle of streakline*: $$x = \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right)\left(12 \text{ s}\right) \qquad (2)$$

and

*last dye particle of streakline*: $$x = \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right)\left(2 \text{ s}\right) \qquad (3)$$

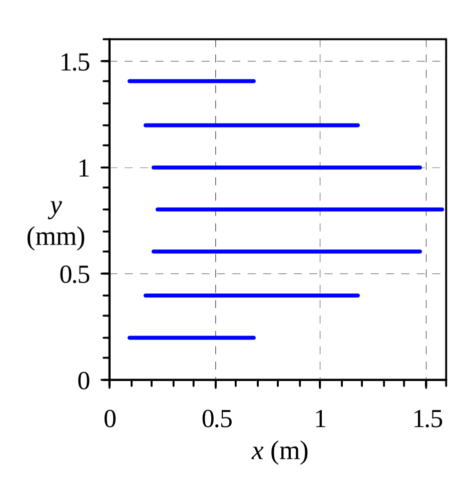



**FIGURE 1**
Streaklines for the given velocity field at $t$ = 12 s. Note that the vertical scale is greatly expanded for clarity ($x$ is in m, but $y$ is in mm).

We substitute the given values of $y$ and the values of $\mu$ and $dP/dx$ into Eqs. 2 and 3 to calculate the ending and beginning $x$ positions of the first released dye particle and the last released dye particle of each streakline. We connect the beginning and ending points to plot the streaklines (**Fig. 1**).

***Discussion*** Both the left and right ends of each dye streak have moved by the same amount compared to those of the previous problem.

---

**4-88** 

**Solution** For a given velocity field we are to compare streaklines at two different times and comment about linear strain rate in the $x$ direction.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Properties*** For water at 40°C, $\mu = 6.53 \times 10^{-4}$ kg/m·s.

***Analysis*** Comparing the results of the previous two problems we see that the streaklines have not stretched at all – they have simply convected downstream. Thus, based on the fundamental definition of linear strain rate, **it is zero**:

*Linear strain rate in the x direction*: $$\boxed{\varepsilon_{xx} = 0} \qquad (1)$$

***Discussion*** Our result agrees with that of Problem 4-83.

---

## 4-89



**Solution** For a given velocity field we are to plot several timelines at a specified time. The timelines are created by hydrogen bubbles released from a vertical wire at $x = 0$.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Properties*** For water at 40°C, $\mu = 6.53\times10^{-4}$ kg/m·s.

***Analysis*** Since the flow is steady, pathlines, streamlines, and streaklines are all straight horizontal lines, but *timelines* are completely different from any of the others. To simulate a timeline, we integrate velocity component *u* with respect to time over the specified time period from $t = 0$ to $t = t_{end}$. We introduce the bubbles at $x = 0$ and at many values of *y* (we used 50 in our simulation). By connecting these *x* locations with a line, we simulate a timeline. The horizontal velocity component is

*x-velocity component*:
$$u = \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right) \qquad (1)$$

We integrate as follows to find the *x* position on the timeline at $t_{end}$:

$$x = x_{start} + \int_{t_{start}}^{t_{end}} u\,dt = 0 + \int_0^{t_{end}}\left(\frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right)\right)dt$$

$$\rightarrow \qquad x = \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right)t_{end}$$

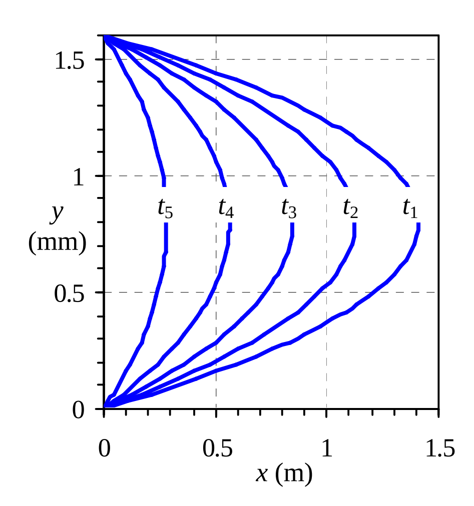



**FIGURE 1**
Timelines for the given velocity field at $t$ = 12.5 s, generated by a simulated hydrogen bubble wire at $x = 0$. Timelines created at $t_5$ = 10.0 s, $t_4$ = 7.5 s, $t_3$ = 5.0 s, $t_2$ = 2.5 s, and $t_1$ = 0 s. Note that the vertical scale is greatly expanded for clarity (*x* is in m, but *y* is in mm).

We substitute the values of *y* and the values of $\mu$ and *dP*/*dx* into the above equation to calculate the ending *x* position of each point in the timeline. We repeat for the five values of $t_{end}$. We plot the timelines in **Fig. 1**.

***Discussion*** Each timeline has the exact shape of the velocity profile.

## 4-90

**Solution** We are to determine if the flow is rotational, and if so calculate the $\theta$-component of vorticity.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is axisymmetric about the *x* axis.

***Analysis*** The velocity components are given by

$$u = \frac{1}{4\mu}\frac{dP}{dx}\left(r^2 - R^2\right) \qquad u_r = 0 \qquad u_\theta = 0 \qquad (1)$$

If the vorticity is non-zero, the flow is rotational. So, we calculate the $\theta$-component of vorticity,

*θ-component of vorticity*:
$$\boxed{\zeta_\theta = \frac{\partial u_r}{\partial z} - \frac{\partial u}{\partial r} = 0 - \frac{1}{4\mu}\frac{dP}{dx}2r = -\frac{r}{2\mu}\frac{dP}{dx}} \qquad (2)$$

Since the vorticity is non-zero, **this flow is rotational**. The vorticity is positive since *dP*/*dx* is negative. In this coordinate system, positive vorticity is counterclockwise with respect to the positive $\theta$ direction. This agrees with our intuition since in the top half of the flow, $\theta$ points out of the page, and the rotation is counterclockwise. Similarly, in the bottom half of the flow, $\theta$ points into the page, and the rotation is clockwise.

***Discussion*** The vorticity varies linearly across the pipe from zero at the centerline to a maximum at the pipe wall.

## 4-91

**Solution** For the given velocity field for axisymmetric Poiseuille flow, we are to calculate the linear strain rates and the shear strain rate.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is axisymmetric about the *x* axis.

***Analysis*** The linear strain rates in the *x* direction and in the *r* direction are

*Linear strain rates*:
$$\varepsilon_{xx} = \frac{\partial u}{\partial x} = \mathbf{0} \qquad \varepsilon_{rr} = \frac{\partial u_r}{\partial r} = \mathbf{0} \qquad (1)$$

Thus there is no linear strain rate in either the *x* or the *r* direction. The shear strain rate in the *x*-*r* plane is

*Shear strain rate*:
$$\boxed{\varepsilon_{xr} = \frac{1}{2}\left(\frac{\partial u_r}{\partial x} + \frac{\partial u}{\partial r}\right) = \frac{1}{2}\left(0 + \frac{1}{4\mu}\frac{dP}{dx}2r\right) = \frac{r}{4\mu}\frac{dP}{dx}} \qquad (2)$$

Fluid particles in this flow have non-zero shear strain rate.

***Discussion*** Since the linear strain rates are zero, fluid particles deform (shear), but do not *stretch* in either the horizontal or radial directions.

## 4-92

**Solution** For the axisymmetric Poiseuille flow velocity field we are to form the axisymmetric strain rate tensor and determine if the *x* and *r* axes are principal axes.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is axisymmetric about the *x* axis.

***Analysis*** The axisymmetric strain rate tensor, $\varepsilon_{ij}$, is

*Axisymmetric strain rate tensor*:
$$\varepsilon_{ij} = \begin{pmatrix} \varepsilon_{rr} & \varepsilon_{rx} \\ \varepsilon_{xr} & \varepsilon_{xx} \end{pmatrix} \qquad (1)$$

We use the linear strain rates and the shear strain rate from the previous problem to generate the tensor,

*Axisymmetric strain rate tensor*:
$$\boxed{\varepsilon_{ij} = \begin{pmatrix} \varepsilon_{rr} & \varepsilon_{rx} \\ \varepsilon_{xr} & \varepsilon_{xx} \end{pmatrix} = \begin{pmatrix} 0 & \dfrac{r}{4\mu}\dfrac{dP}{dx} \\ \dfrac{r}{4\mu}\dfrac{dP}{dx} & 0 \end{pmatrix}} \qquad (2)$$

Note that by symmetry $\varepsilon_{rx} = \varepsilon_{xr}$. If the *x* and *r* axes were principal axes, the diagonals of $\varepsilon_{ij}$ would be non-zero, and the off-diagonals would be zero. Here we have the opposite case, so **the *x* and *r* axes are not principal axes**.

***Discussion*** The principal axes can be calculated using tensor algebra.

**4-93**

**Solution** We are to determine the location of stagnation point(s) in a given velocity field.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The velocity components are

*x-component of velocity*:
$$u = \frac{-\dot{V}x}{\pi L}\frac{x^2+y^2+b^2}{x^4+2x^2y^2+2x^2b^2+y^4-2y^2b^2+b^4} \qquad (1)$$

and

*y-component of velocity*:
$$v = \frac{-\dot{V}y}{\pi L}\frac{x^2+y^2-b^2}{x^4+2x^2y^2+2x^2b^2+y^4-2y^2b^2+b^4} \qquad (2)$$

Both *u* and *v* must be zero at a stagnation point. From Eq. 1, *u* can be zero only when $x = 0$. From Eq. 2, *v* can be zero either when $y = 0$ or when $x^2 + y^2 - b^2 = 0$. Combining the former with the result from Eq. 1, we see that **there is a stagnation point at (*x*,*y*) = (0,0), i.e. at the origin**,

*Stagnation point*:
$$\boxed{u = 0 \text{ and } v = 0 \text{ at } (x, y) = (0,0)} \qquad (3)$$

Combining the latter with the result from Eq. 1, there appears to be another stagnation point at (*x*,*y*) = (0,*b*). However, at that location, Eq. 2 becomes

*y-component of velocity*:
$$v = \frac{-\dot{V}b}{\pi L}\frac{0}{b^4-2b^2b^2+b^4} = \frac{0}{0} \qquad (4)$$

This point turns out to be a **singularity point** in the flow. Thus, **the location (0,*b*) is *not* a stagnation point** after all.

***Discussion*** There is only one stagnation point in this flow, and it is at the origin.

## 4-94

**Solution** We are to draw a velocity vector plot for a given velocity field.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** We generate an array of *x* and *y* values in the given range and calculate *u* and *v* from Eqs. 1 and 2 respectively at each location. We choose an appropriate scale factor for the vectors and then draw arrows to form the velocity vector plot (**Fig. 1**).

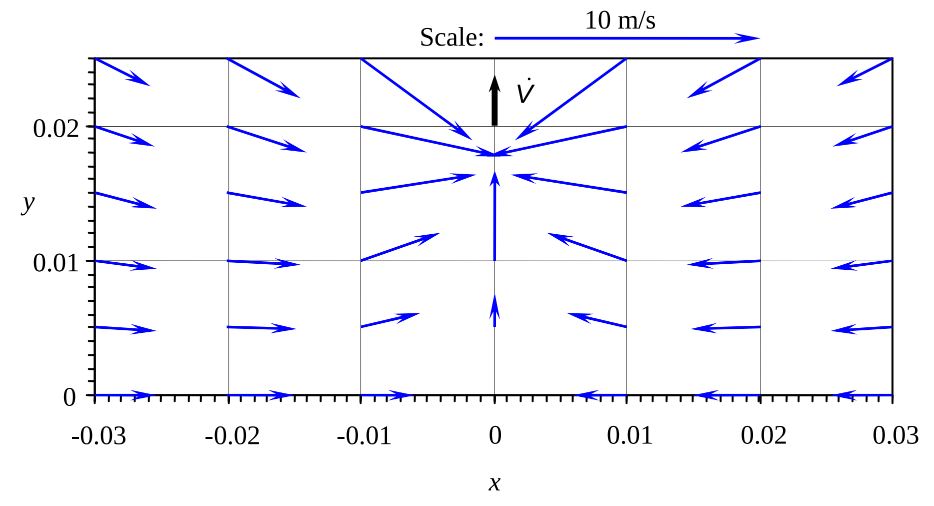



**FIGURE 1**
Velocity vector plot for the vacuum cleaner; the scale factor for the velocity vectors is shown on the legend. *x* and *y* values are in meters. The vacuum cleaner inlet is at the point *x* = 0, *y* = 0.02 m.

It is clear from the velocity vector plot how the air gets sucked into the vacuum cleaner from all directions. We also see that there is no flow through the floor.

***Discussion*** We discuss this problem in more detail in Chap. 10.

---

## 4-95

**Solution** We are to calculate the speed of air along the floor due to a vacuum cleaner, and find the location of maximum speed.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** At the floor, *y* = 0. Setting *y* = 0 in Eq. 2 of Problem 4-93 shows that *v* = 0, as expected – no flow through the floor. Setting *y* = 0 in Eq. 1 of Problem 4-93 results in the speed along the floor,

*Speed on the floor*:

$$u = \frac{-\dot{V}x}{\pi L}\frac{x^2+b^2}{x^4+2x^2b^2+b^4} = \frac{-\dot{V}x}{\pi L}\frac{x^2+b^2}{\left(x^2+b^2\right)^2} = \frac{-\dot{V}x}{\pi L\left(x^2+b^2\right)} \qquad (1)$$

We find the maximum speed be differentiating Eq. 1 and setting the result to zero,

*Maximum speed on the floor*:

$$\frac{du}{dx} = \frac{-\dot{V}}{\pi L}\left[\frac{-2x^2}{\left(x^2+b^2\right)^2} + \frac{1}{x^2+b^2}\right] = 0 \qquad (2)$$

After some algebraic manipulation, we find that Eq. 2 has solutions at *x* = *b* and *x* = -*b*. **It is at *x* = *b* and *x* = -*b* where we expect the best performance**. At the origin, directly below the vacuum cleaner inlet, the flow is stagnant. Thus, despite our intuition, **the vacuum cleaner will work poorly directly below the inlet**.

***Discussion*** Try some experiments at home to verify these results!

**4-96**

**Solution** For a given expression for *u*, we are to find an expression for *v* such that the flow field is incompressible.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The *x*-component of velocity is given as

*x-component of velocity*: $$u = a + b\left(x - c\right)^2 \qquad (1)$$

In order for the flow field to be incompressible, the volumetric strain rate must be zero,

*Volumetric strain rate*: $$\frac{1}{V}\frac{DV}{Dt} = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \underbrace{\cancel{\frac{\partial w}{\partial z}}}_{\text{Two-D}} = 0 \qquad (2)$$

This gives us a necessary condition for *v*,

*Necessary condition for v*: $$\frac{\partial v}{\partial y} = -\frac{\partial u}{\partial x} \qquad (3)$$

We substitute Eq. 1 into Eq. 3 and integrate to solve for *v*,

*Expression for v*:
$$\frac{\partial v}{\partial y} = -\frac{\partial u}{\partial x} = -2b\left(x - c\right)$$
$$v = \int \frac{\partial v}{\partial y} dy = \int \left(-2b\left(x - c\right)\right) dy + f(x)$$

Note that we must add an arbitrary function of *x* rather than a simple constant of integration since this is a partial integration with respect to *y*. *v* is a function of both *x* and *y*. The result of the integration is

*Expression for v*: $$\boxed{v = -2b\left(x - c\right)y + f(x)} \qquad (4)$$

***Discussion*** We verify by plugging Eqs. 1 and 4 into Eq. 2,

*Volumetric strain rate*: $$\frac{1}{V}\frac{DV}{Dt} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 2b\left(x - c\right) - 2b\left(x - c\right) = 0 \qquad (5)$$

Since the volumetric strain rate is zero for any function *f*(*x*), Eqs. 1 and 4 represent an incompressible flow field.

## 4-97

**Solution** For a given velocity field we are to determine if the flow is rotational or irrotational.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis*** The velocity components for flow over a circular cylinder of radius $r$ are

$$u_r = V\cos\theta\left(1-\frac{a^2}{r^2}\right) \qquad u_\theta = -V\sin\theta\left(1+\frac{a^2}{r^2}\right) \tag{1}$$

Since the flow is assumed to be two-dimensional in the $r$-$\theta$ plane, the only non-zero component of vorticity is in the $z$ direction. In cylindrical coordinates,

*Vorticity component in the z direction*:

$$\zeta_z = \frac{1}{r}\left(\frac{\partial(ru_\theta)}{\partial r} - \frac{\partial u_r}{\partial\theta}\right) \tag{2}$$

We plug in the velocity components of Eq. 1 into Eq. 2 to solve for $\zeta_z$,

$$\zeta_z = \frac{1}{r}\left[\frac{\partial}{\partial r}\left(-V\sin\theta\left(r+\frac{a^2}{r}\right)\right)+V\sin\theta\left(1-\frac{a^2}{r^2}\right)\right] = \frac{1}{r}\left[-V\sin\theta+V\frac{a^2}{r^2}\sin\theta+V\sin\theta-V\frac{a^2}{r^2}\sin\theta\right]=0 \tag{3}$$

Hence, since the vorticity is everywhere zero, **this flow is irrotational**.

***Discussion*** Fluid particles distort as they flow around the cylinder, but their net rotation is zero.

---

## 4-98

**Solution** For a given velocity field we are to find the location of the stagnation point.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis*** The stagnation point occurs when both components of velocity are zero. We set $u_r$ = 0 and $u_\theta$ = 0 in Eq. 1 of the previous problem,

$$u_r = V\cos\theta\left(1-\frac{a^2}{r^2}\right)=0 \qquad \text{Either } \cos\theta = 0 \text{ or } r^2 = a^2$$
$$u_\theta = -V\sin\theta\left(1+\frac{a^2}{r^2}\right)=0 \qquad \text{Either } \sin\theta = 0 \text{ or } r^2 = -a^2 \tag{1}$$

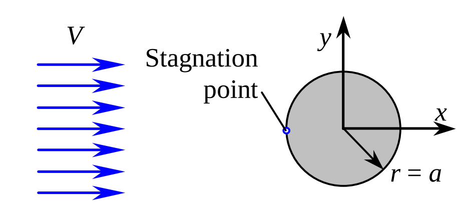



**FIGURE 1**
The stagnation point on the upstream half of the flow field is located at the nose of the cylinder at $r = a$ and $\theta = 180^\circ$.

The second part of the $u_\theta$ condition in Eq. 1 is obviously impossible since cylinder radius $a$ is a real number. Thus $\sin\theta = 0$, which means that $\theta = 0^\circ$ or $180^\circ$. We are restricted to the left half of the flow ($x < 0$); therefore we choose $\theta = 180^\circ$. Now we look at the $u_r$ condition in Eq. 1. At $\theta = 180^\circ$, $\cos\theta$ = -1, and thus we conclude that $r$ must equal $a$. Summarizing,

*Stagnation point*:

$$\boxed{r = a \qquad \theta = -180^\circ} \tag{2}$$

Or, in Cartesian coordinates,

*Stagnation point*:

$$\boxed{x = -a \qquad y = 0} \tag{3}$$

The stagnation point is located at the nose of the cylinder (Fig. 1).

***Discussion*** This result agrees with our intuition, since the fluid must divert around the cylinder at the nose.

---

## 4-99



**Solution** For a given stream function we are to generate an equation for streamlines, and then plot several streamlines in the upstream half of the flow field.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis***
($a$) The stream function is

$$\psi = V\sin\theta\left(r - \frac{a^2}{r}\right) \quad (1)$$

First we multiply both sides of Eq. 1 by $r$, and then solve the quadratic equation for $r$ using the quadratic rule. This gives us an equation for $r$ as a function of $\theta$, with $\psi$, $a$, and $V$ as parameters,

*Equation for a streamline*:

$$\boxed{r = \frac{\psi \pm \sqrt{\psi^2 + 4a^2V^2\sin^2\theta}}{2V\sin\theta}} \quad (2)$$

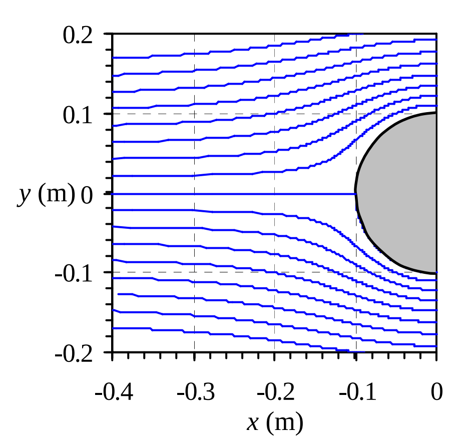



**FIGURE 1**
Streamlines corresponding to flow over a circular cylinder. Only the upstream half of the flow field is plotted.

($b$) For the particular case in which $V$ = 1.00 m/s and cylinder radius $a$ = 10.0 cm, we choose various values of $\psi$ in Eq. 2, and plot streamlines in the upstream half of the flow (**Fig. 1**). Each value of $\psi$ corresponds to a unique streamline.

***Discussion*** The stream function is discussed in greater detail in Chap. 9.

---

## 4-100

**Solution** For a given velocity field we are to calculate the linear strain rates $\varepsilon_{rr}$ and $\varepsilon_{\theta\theta}$ in the $r$-$\theta$ plane.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis*** We substitute the equation of Problem 4-97 into that of Problem 4-91,

*Linear strain rate in r direction*:

$$\boxed{\varepsilon_{rr} = \frac{\partial u_r}{\partial r} = 2V\cos\theta\frac{a^2}{r^3}} \quad (1)$$

and

*Linear strain rate in $\theta$ direction*:

$$\boxed{\varepsilon_{\theta\theta} = \frac{1}{r}\left[\frac{\partial u_\theta}{\partial\theta} + u_r\right] = \frac{1}{r}\left[-V\cos\theta\left(1+\frac{a^2}{r^2}\right) + V\cos\theta\left(1-\frac{a^2}{r^2}\right)\right] = -2V\cos\theta\frac{a^2}{r^3}} \quad (2)$$

The linear strain rates are non-zero, implying that fluid line segments *do* stretch (or shrink) as they move about in the flow field.

***Discussion*** The linear strain rates decrease rapidly with distance from the cylinder.

---

## 4-101

**Solution** We are to discuss whether the flow field of the previous problem is incompressible or compressible.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis*** For two-dimensional flow we know that a flow is incompressible if its volumetric strain rate is zero. In that case,

*Volumetric strain rate, incompressible 2-D flow in the x-y plane*:
$$\frac{1}{V}\frac{DV}{Dt} = \varepsilon_{xx} + \varepsilon_{yy} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad (1)$$

We can extend Eq. 1 to cylindrical coordinates by writing

*Volumetric strain rate, incompressible 2-D flow in the r-θ plane*:
$$\frac{1}{V}\frac{DV}{Dt} = \varepsilon_{rr} + \varepsilon_{r\theta} = \frac{\partial u_r}{\partial r} + \frac{1}{r}\left[\frac{\partial u_\theta}{\partial \theta} + u_r\right] = 0 \qquad (2)$$

Plugging in the results of the previous problem we see that

*Volumetric strain rate for flow over a circular cylinder*:
$$\frac{1}{V}\frac{DV}{Dt} = 2V\cos\theta\frac{a^2}{r^3} - 2V\cos\theta\frac{a^2}{r^3} = 0 \qquad (3)$$

Since the volumetric strain rate is zero everywhere, **the flow is incompressible**.

***Discussion*** In Chap. 9 we show that Eq. 2 can be obtained from the differential equation for conservation of mass.

---

## 4-102

**Solution** For a given velocity field we are to calculate the shear strain rate $\varepsilon_{r\theta}$.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis*** We substitute the equation of Problem 4-97 into that of Problem 4-91,

*Shear strain rate in r-θ plane*:
$$\begin{aligned}\varepsilon_{r\theta} &= \frac{1}{2}\left[r\frac{\partial}{\partial r}\left(\frac{u_\theta}{r}\right) + \frac{1}{r}\frac{\partial u_r}{\partial \theta}\right] \\ &= \frac{1}{2}\left[r\frac{\partial}{\partial r}\left(-\frac{V\sin\theta}{r} - V\sin\theta\frac{a^2}{r^3}\right) + \frac{1}{r}\left(-V\sin\theta\left(1-\frac{a^2}{r^2}\right)\right)\right] \qquad (1) \\ &= \frac{1}{2}V\sin\theta\left[\frac{1}{r} + 3\frac{a^2}{r^3} - \frac{1}{r} + \frac{a^2}{r^3}\right] = 2V\sin\theta\frac{a^2}{r^3}\end{aligned}$$

which reduces to

*Shear strain rate in r-θ plane*:
$$\boxed{\varepsilon_{r\theta} = 2V\sin\theta\frac{a^2}{r^3}} \qquad (2)$$

The shear strain rate is non-zero, implying that fluid line segments *do* deform with shear as they move about in the flow field.

***Discussion*** The shear strain rate decreases rapidly (as $r^{-3}$) with distance from the cylinder.

---



# Solutions Manual for

# Fluid Mechanics: Fundamentals and Applications

# by Çengel & Cimbala

---

# CHAPTER 5

# MASS, BERNOULLI, AND ENERGY EQUATIONS

---

## Conservation of Mass

**5-1C**

**Solution** We are to name some conserved and non-conserved quantities.

***Analysis*** **Mass, energy, momentum, and electric charge are conserved**, and **volume and entropy are not conserved** during a process.

***Discussion*** Students may think of other answers that may be equally valid.

**5-2C**

**Solution** We are to discuss mass and volume flow rates and their relationship.

***Analysis*** *Mass flow rate* is the **amount of mass flowing through a cross-section per unit time** whereas *volume flow rate* is the **amount of volume flowing through a cross-section per unit time**.

***Discussion*** Mass flow rate has dimensions of mass/time while volume flow rate has dimensions of volume/time.

**5-3C**

**Solution** We are to discuss the mass flow rate entering and leaving a control volume.

***Analysis*** The amount of mass or energy entering a control volume **does not have to be equal** to the amount of mass or energy leaving during an unsteady-flow process.

***Discussion*** If the process is steady, however, the two mass flow rates must be equal; otherwise the amount of mass would have to increase or decrease inside the control volume, which would make it unsteady.

**5-4C**

**Solution** We are to discuss steady flow through a control volume.

***Analysis*** Flow through a control volume is *steady* when it involves **no changes with time at any specified position**.

***Discussion*** This applies to any variable we might consider – pressure, velocity, density, temperature, etc.

**5-5C**

**Solution** We are to discuss whether the flow is steady through a given control volume.

***Analysis*** **No**, a flow with the same volume flow rate at the inlet and the exit is **not necessarily steady** (unless the density is constant). To be steady, the mass flow rate through the device must remain constant.

***Discussion*** If the question had stated that the two *mass* flow rates were equal, then the answer would be *yes*.

**5-6E**

**Solution** A garden hose is used to fill a water bucket. The volume and mass flow rates of water, the filling time, and the discharge velocity are to be determined.

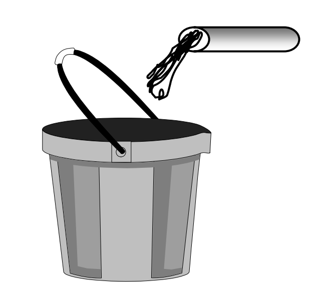
***Assumptions*** **1** Water is an incompressible substance. **2** Flow through the hose is steady. **3** There is no waste of water by splashing.

***Properties*** We take the density of water to be 62.4 lbm/ft$^3$.

***Analysis*** (*a*) The volume and mass flow rates of water are

$$\dot{V} = AV = (\pi D^2/4)V = [\pi(1/12 \text{ ft})^2/4](8 \text{ ft/s}) = 0.04363 \text{ ft}^3\text{/s} \cong \mathbf{0.0436\ ft^3/s}$$

$$\dot{m} = \rho\dot{V} = (62.4 \text{ lbm/ft}^3)(0.04363 \text{ ft}^3\text{/s}) = \mathbf{2.72\ lbm/s}$$



(*b*) The time it takes to fill a 20-gallon bucket is

$$\Delta t = \frac{V}{\dot{V}} = \frac{20 \text{ gal}}{0.04363 \text{ ft}^3\text{/s}}\left(\frac{1 \text{ ft}^3}{7.4804 \text{ gal}}\right) = \mathbf{61.3\ s}$$

(*c*) The average discharge velocity of water at the nozzle exit is

$$V_e = \frac{\dot{V}}{A_e} = \frac{\dot{V}}{\pi D_e^2/4} = \frac{0.04363 \text{ ft}^3\text{/s}}{[\pi(0.5/12 \text{ ft})^2/4]} = \mathbf{32\ ft/s}$$

***Discussion*** Note that for a given flow rate, the average velocity is inversely proportional to the square of the velocity. Therefore, when the diameter is reduced by half, the velocity quadruples.

---

**5-7**

**Solution** Air is accelerated in a nozzle. The mass flow rate and the exit area of the nozzle are to be determined.

***Assumptions*** Flow through the nozzle is steady.

***Properties*** The density of air is given to be 2.21 kg/m$^3$ at the inlet, and 0.762 kg/m$^3$ at the exit.

***Analysis*** (*a*) The mass flow rate of air is determined from the inlet conditions to be

$$\dot{m} = \rho_1 A_1 V_1 = (2.21 \text{ kg/m}^3)(0.008 \text{ m}^2)(30 \text{ m/s}) = \mathbf{0.530\ kg/s}$$

$V_1$ = 30 m/s
$A_1$ = 80 cm$^2$
AIR
$V_2$ = 180 m/s

(*b*) There is only one inlet and one exit, and thus $\dot{m}_1 = \dot{m}_2 = \dot{m}$. Then the exit area of the nozzle is determined to be

$$\dot{m} = \rho_2 A_2 V_2 \longrightarrow A_2 = \frac{\dot{m}}{\rho_2 V_2} = \frac{0.530 \text{ kg/s}}{(0.762 \text{ kg/m}^3)(180 \text{ m/s})} = 0.00387 \text{ m}^2 = \mathbf{38.7\ cm^2}$$

***Discussion*** Since this is a compressible flow, we must equate mass flow rates, not volume flow rates.

---

**5-8**

**Solution** Air is expanded and is accelerated as it is heated by a hair dryer of constant diameter. The percent increase in the velocity of air as it flows through the drier is to be determined.

***Assumptions*** Flow through the nozzle is steady.

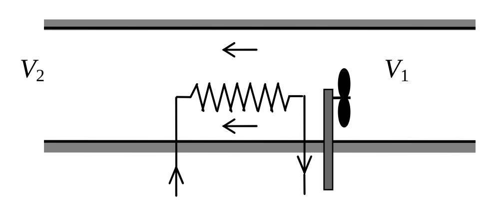
***Properties*** The density of air is given to be 1.20 kg/m$^3$ at the inlet, and 1.05 kg/m$^3$ at the exit.

***Analysis*** There is only one inlet and one exit, and thus $\dot{m}_1 = \dot{m}_2 = \dot{m}$. Then,

$$\dot{m}_1 = \dot{m}_2$$

$$\rho_1 A V_1 = \rho_2 A V_2$$

$$\frac{V_2}{V_1} = \frac{\rho_1}{\rho_2} = \frac{1.20 \text{ kg/m}^3}{1.05 \text{ kg/m}^3} = 1.14 \quad \text{(or, an increase of } \mathbf{14\%}\text{)}$$




Therefore, the air velocity increases **14%** as it flows through the hair drier.

***Discussion*** It makes sense that the velocity *increases* since the density *decreases*, but the mass flow rate is constant.

---

**5-9E**

**Solution** The ducts of an air-conditioning system pass through an open area. The inlet velocity and the mass flow rate of air are to be determined.

***Assumptions*** Flow through the air conditioning duct is steady.

***Properties*** The density of air is given to be 0.078 lbm/ft$^3$ at the inlet.

***Analysis*** The inlet velocity of air and the mass flow rate through the duct are

$$V_1 = \frac{\dot{V}_1}{A_1} = \frac{\dot{V}_1}{\pi D^2/4} = \frac{450 \text{ ft}^3\text{/min}}{\pi (10/12 \text{ ft})^2/4} = \mathbf{825 \text{ ft/min}} = \mathbf{13.8 \text{ ft/s}}$$

450 ft$^3$/min AIR $D$ = 10 in

$$\dot{m} = \rho_1 \dot{V}_1 = (0.078 \text{ lbm/ft}^3)(450 \text{ ft}^3\text{/min}) = 35.1 \text{ lbm/min} = \mathbf{0.585 \text{ lbm/s}}$$

***Discussion*** The mass flow rate though a duct must remain constant in steady flow; however, the volume flow rate varies since the density varies with the temperature and pressure in the duct.

---

**5-10**

**Solution** A rigid tank initially contains air at atmospheric conditions. The tank is connected to a supply line, and air is allowed to enter the tank until the density rises to a specified level. The mass of air that entered the tank is to be determined.

***Properties*** The density of air is given to be 1.18 kg/m$^3$ at the beginning, and 7.20 kg/m$^3$ at the end.

***Analysis*** We take the tank as the system, which is a control volume since mass crosses the boundary. The mass balance for this system can be expressed as

*Mass balance*: $m_{in} - m_{out} = \Delta m_{\text{system}} \;\rightarrow\; m_i = m_2 - m_1 = \rho_2 V - \rho_1 V$

Substituting, $m_i = (\rho_2 - \rho_1)V = [(7.20 - 1.18) \text{ kg/m}^3](1 \text{ m}^3) = \mathbf{6.02 \text{ kg}}$

$V_1$ = 1 m$^3$
$\rho_1$ = 1.18 kg/m$^3$

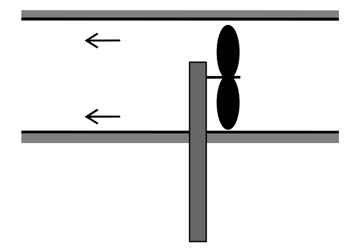
Therefore, **6.02 kg of mass entered the tank**.

***Discussion*** Tank temperature and pressure do not enter into the calculations.

---

**5-11**

**Solution** The ventilating fan of the bathroom of a building runs continuously. The mass of air "vented out" per day is to be determined.



***Assumptions*** Flow through the fan is steady.

***Properties*** The density of air in the building is given to be 1.20 kg/m$^3$.

***Analysis*** The mass flow rate of air vented out is

$$\dot{m}_{\text{air}} = \rho \dot{V}_{\text{air}} = (1.20 \text{ kg/m}^3)(0.030 \text{ m}^3/\text{s}) = 0.036 \text{ kg/s}$$

Then the mass of air vented out in 24 h becomes

$$m = \dot{m}_{\text{air}} \Delta t = (0.036 \text{ kg/s})(24 \times 3600 \text{ s}) = \mathbf{3110 \ kg}$$

***Discussion*** Note that more than 3 tons of air is vented out by a bathroom fan in one day.

---

**5-12**

**Solution** A desktop computer is to be cooled by a fan at a high elevation where the air density is low. The mass flow rate of air through the fan and the diameter of the casing for a given velocity are to be determined.



***Assumptions*** Flow through the fan is steady.

***Properties*** The density of air at a high elevation is given to be 0.7 kg/m$^3$.

***Analysis*** The mass flow rate of air is

$$\dot{m}_{\text{air}} = \rho \dot{V}_{\text{air}} = (0.7 \text{ kg/m}^3)(0.34 \text{ m}^3/\text{min}) = 0.238 \text{ kg/min} = \mathbf{0.0040 \ kg/s}$$

If the mean velocity is 110 m/min, the diameter of the casing is

$$\dot{V} = AV = \frac{\pi D^2}{4} V \quad \rightarrow \quad D = \sqrt{\frac{4\dot{V}}{\pi V}} = \sqrt{\frac{4(0.34 \text{ m}^3/\text{min})}{\pi (110 \text{ m/min})}} = \mathbf{0.063 \ m}$$

Therefore, the diameter of the casing must be **at least 6.3 cm** to ensure that the mean velocity does not exceed 110 m/min.

***Discussion*** This problem shows that engineering systems are sized to satisfy given imposed constraints.

---

**5-13**

**Solution** A smoking lounge that can accommodate 15 smokers is considered. The required minimum flow rate of air that needs to be supplied to the lounge and the diameter of the duct are to be determined.

***Assumptions*** Infiltration of air into the smoking lounge is negligible.

***Properties*** The minimum fresh air requirements for a smoking lounge is given to be 30 L/s per person.

***Analysis*** The required minimum flow rate of air that needs to be supplied to the lounge is determined directly from

$$\dot{V}_{\text{air}} = \dot{V}_{\text{air per person}} (\text{No. of persons})$$
$$= (30 \text{ L/s} \cdot \text{person})(15 \text{ persons}) = 450 \text{ L/s} = \mathbf{0.45 \ m^3/s}$$

Smoking Lounge

15 smokers
30 L/s person

The volume flow rate of fresh air can be expressed as

$$\dot{V} = VA = V(\pi D^2/4)$$

Solving for the diameter $D$ and substituting,

$$D = \sqrt{\frac{4\dot{V}}{\pi V}} = \sqrt{\frac{4(0.45 \text{ m}^3/\text{s})}{\pi (8 \text{ m/s})}} = \mathbf{0.268 \ m}$$

Therefore, the diameter of the fresh air duct should be **at least 26.8 cm** if the velocity of air is not to exceed 8 m/s.

***Discussion*** Fresh air requirements in buildings must be taken seriously to avoid health problems.

---

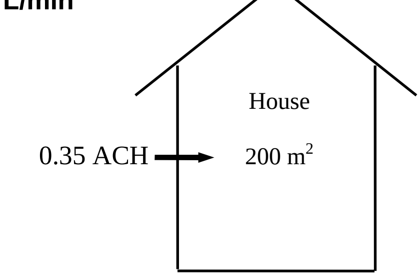
**5-14**

**Solution** The minimum fresh air requirements of a residential building is specified to be 0.35 air changes per hour. The size of the fan that needs to be installed and the diameter of the duct are to be determined.

***Analysis*** The volume of the building and the required minimum volume flow rate of fresh air are

$$V_{\text{room}} = (2.7\text{ m})(200\text{ m}^2) = 540\text{ m}^3$$

$$\dot{V} = V_{\text{room}} \times \text{ACH} = (540\text{ m}^3)(0.35/\text{h}) = 189\text{ m}^3/\text{h} = 189{,}000\text{ L/h} = \mathbf{3150\ L/min}$$




The volume flow rate of fresh air can be expressed as

$$\dot{V} = VA = V(\pi D^2/4)$$

Solving for the diameter *D* and substituting,

$$D = \sqrt{\frac{4\dot{V}}{\pi V}} = \sqrt{\frac{4(189/3600\text{ m}^3/\text{s})}{\pi(6\text{ m/s})}} = \mathbf{0.106\ m}$$

Therefore, the diameter of the fresh air duct should be **at least 10.6 cm** if the velocity of air is not to exceed 6 m/s.

***Discussion*** Fresh air requirements in buildings must be taken seriously to avoid health problems.

## Mechanical Energy and Pump Efficiency

**5-15C**

**Solution** We are to discuss mechanical energy and how it differs from thermal energy.

***Analysis*** *Mechanical energy* is the **form of energy that can be converted to mechanical work completely and directly by a mechanical device** such as a propeller. It differs from thermal energy in that **thermal energy cannot be converted to work directly and completely**. The forms of mechanical energy of a fluid stream are kinetic, potential, and flow energies.

***Discussion*** It would be nice if we could convert thermal energy completely into work. However, this would violate the second law of thermodynamics.

---

**5-16C**

**Solution** We are to define and discuss mechanical efficiency.

***Analysis*** *Mechanical efficiency* is defined as **the ratio of the mechanical energy output to the mechanical energy input**. A mechanical efficiency of 100% for a hydraulic turbine means that the entire mechanical energy of the fluid is converted to mechanical (shaft) work.

***Discussion*** No real fluid machine is 100% efficient, due to frictional losses, etc. – the second law of thermodynamics.

---

**5-17C**

**Solution** We are to define and discuss pump-motor efficiency.

***Analysis*** The *combined pump-motor efficiency* of a pump/motor system is defined as the ratio of the increase in the mechanical energy of the fluid to the electrical power consumption of the motor,

$$\eta_{\text{pump-motor}} = \eta_{\text{pump}}\eta_{\text{motor}} = \frac{\dot{E}_{\text{mech,out}} - \dot{E}_{\text{mech,in}}}{\dot{W}_{\text{elect,in}}} = \frac{\Delta\dot{E}_{\text{mech,fluid}}}{\dot{W}_{\text{elect,in}}} = \frac{\dot{W}_{\text{pump}}}{\dot{W}_{\text{elect,in}}}$$

The combined pump-motor efficiency cannot be greater than either of the pump or motor efficiency since both pump and motor efficiencies are less than 1, and the product of two numbers that are less than one is less than either of the numbers.

***Discussion*** Since many pumps are supplied with an integrated motor, pump-motor efficiency is a useful parameter.

---

**5-18C**

**Solution** We are to define and discuss turbine, generator, and turbine-generator efficiency.

***Analysis*** Turbine efficiency, generator efficiency, and *combined turbine-generator efficiency* are defined as follows:

$$\eta_{\text{turbine}} = \frac{\text{Mechanical energy output}}{\text{Mechanical energy extracted from the fluid}} = \frac{\dot{W}_{\text{shaft,out}}}{|\Delta\dot{E}_{\text{mech,fluid}}|}$$

$$\eta_{\text{generator}} = \frac{\text{Electrical power output}}{\text{Mechanical power input}} = \frac{\dot{W}_{\text{elect,out}}}{\dot{W}_{\text{shaft,in}}}$$

$$\eta_{\text{turbine-gen}} = \eta_{\text{turbine}}\eta_{\text{generaor}} = \frac{\dot{W}_{\text{elect,out}}}{\dot{E}_{\text{mech,in}} - \dot{E}_{\text{mech,out}}} = \frac{\dot{W}_{\text{elect,out}}}{|\Delta\dot{E}_{\text{mech,fluid}}|}$$

***Discussion*** Most turbines are connected directly to a generator, so the combined efficiency is a useful parameter.

---

**5-19**

**Solution** A river is flowing at a specified velocity, flow rate, and elevation. The total mechanical energy of the river water per unit mass, and the power generation potential of the entire river are to be determined.

***Assumptions*** **1** The elevation given is the elevation of the free surface of the river. **2** The velocity given is the average velocity. **3** The mechanical energy of water at the turbine exit is negligible.

***Properties*** We take the density of water to be $\rho = 1000\ \text{kg/m}^3$.

***Analysis*** Noting that the sum of the flow energy and the potential energy is constant for a given fluid body, we can take the elevation of the entire river water to be the elevation of the free surface, and ignore the flow energy. Then the total mechanical energy of the river water per unit mass becomes

$$e_{\text{mech}} = pe + ke = gh + \frac{V^2}{2}$$

$$= \left((9.81\ \text{m/s}^2)(90\ \text{m}) + \frac{(3\ \text{m/s})^2}{2}\right)\left(\frac{1\ \text{kJ/kg}}{1000\ \text{m}^2/\text{s}^2}\right)$$

$$= 0.887\ \text{kJ/kg}$$

River

3 m/s

90 m

The power generation potential of the river water is obtained by multiplying the total mechanical energy by the mass flow rate,

$$\dot{m} = \rho\dot{V} = (1000\ \text{kg/m}^3)(500\ \text{m}^3/\text{s}) = 500{,}000\ \text{kg/s}$$

$$\dot{W}_{\max} = \dot{E}_{\text{mech}} = \dot{m}e_{\text{mech}} = (500{,}000\ \text{kg/s})(0.887\ \text{kg/s}) = 444{,}000\ \text{kW} = \mathbf{444\ MW}$$

Therefore, 444 MW of power can be generated from this river as it discharges into the lake if its power potential can be recovered completely.

***Discussion*** Note that the kinetic energy of water is negligible compared to the potential energy, and it can be ignored in the analysis. Also, the power output of an actual turbine will be less than 444 MW because of losses and inefficiencies.

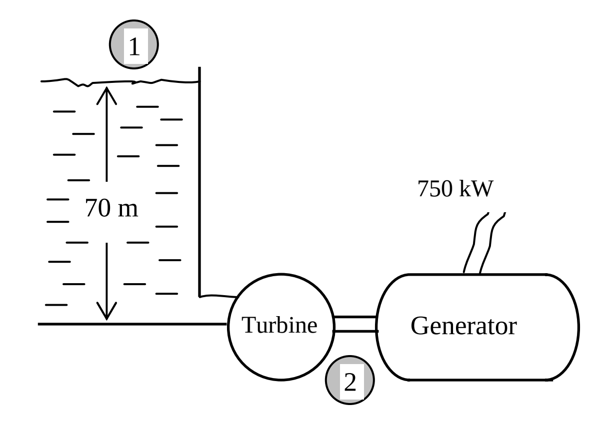
**5-20**

**Solution** A hydraulic turbine-generator is generating electricity from the water of a large reservoir. The combined turbine-generator efficiency and the turbine efficiency are to be determined.

***Assumptions*** **1** The elevation of the reservoir remains constant. **2** The mechanical energy of water at the turbine exit is negligible.

***Analysis*** We take the free surface of the reservoir to be point 1 and the turbine exit to be point 2. We also take the turbine exit as the reference level ($z_2 = 0$), and thus the potential energy at points 1 and 2 are $\text{pe}_1 = gz_1$ and $\text{pe}_2 = 0$. The flow energy $P/\rho$ at both points is zero since both 1 and 2 are open to the atmosphere ($P_1 = P_2 = P_{\text{atm}}$). Further, the kinetic energy at both points is zero ($\text{ke}_1 = \text{ke}_2 = 0$) since the water at point 1 is essentially motionless, and the kinetic energy of water at turbine exit is assumed to be negligible. The potential energy of water at point 1 is

$$pe_1 = gz_1 = (9.81\,\text{m/s}^2)(70\,\text{m})\left(\frac{1\,\text{kJ/kg}}{1000\,\text{m}^2/\text{s}^2}\right) = 0.687\,\text{kJ/kg}$$




Then the rate at which the mechanical energy of the fluid is supplied to the turbine become

$$\left|\Delta\dot{E}_{\text{mech,fluid}}\right| = \dot{m}(e_{\text{mech,in}} - e_{\text{mech,out}}) = \dot{m}(pe_1 - 0) = \dot{m}pe_1$$
$$= (1500\,\text{kg/s})(0.687\,\text{kJ/kg})$$
$$= 1031\,\text{kW}$$

The combined turbine-generator and the turbine efficiency are determined from their definitions,

$$\eta_{\text{turbine-gen}} = \frac{\dot{W}_{\text{elect,out}}}{|\Delta\dot{E}_{\text{mech,fluid}}|} = \frac{750\,\text{kW}}{1031\,\text{kW}} = 0.727 \quad \text{or} \quad \mathbf{72.7\%}$$

$$\eta_{\text{turbine}} = \frac{\dot{W}_{\text{shaft,out}}}{|\Delta\dot{E}_{\text{mech,fluid}}|} = \frac{800\,\text{kW}}{1031\,\text{kW}} = 0.776 \quad \text{or} \quad \mathbf{77.6\%}$$

Therefore, the reservoir supplies 1031 kW of mechanical energy to the turbine, which converts 800 kW of it to shaft work that drives the generator, which generates 750 kW of electric power.

***Discussion*** This problem can also be solved by taking point 1 to be at the turbine inlet, and using flow energy instead of potential energy. It would give the same result since the flow energy at the turbine inlet is equal to the potential energy at the free surface of the reservoir.

**5-21**

**Solution** Wind is blowing steadily at a certain velocity. The mechanical energy of air per unit mass, the power generation potential, and the actual electric power generation are to be determined.

***Assumptions*** **1** The wind is blowing steadily at a constant uniform velocity. **2** The efficiency of the wind turbine is independent of the wind speed.

***Properties*** The density of air is given to be $\rho = 1.25$ kg/m$^3$.

***Analysis*** Kinetic energy is the only form of mechanical energy the wind possesses, and it can be converted to work entirely. Therefore, the power potential of the wind is its kinetic energy, which is $V^2/2$ per unit mass, and $\dot{m}V^2/2$ for a given mass flow rate:

$$e_{\text{mech}} = ke = \frac{V^2}{2} = \frac{(12\text{ m/s})^2}{2}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 0.072\text{ kJ/kg}$$

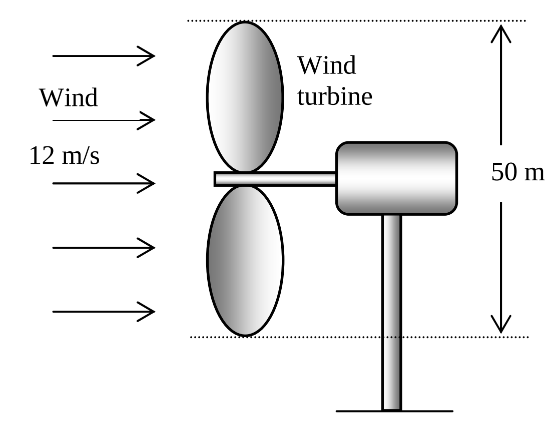
$$\dot{m} = \rho VA = \rho V\frac{\pi D^2}{4} = (1.25\text{ kg/m}^3)(12\text{ m/s})\frac{\pi(50\text{ m})^2}{4} = 29{,}452\text{ kg/s}$$

$$\dot{W}_{\max} = \dot{E}_{\text{mech}} = \dot{m}e_{\text{mech}} = (29{,}452\text{ kg/s})(0.072\text{ kJ/kg}) = 2121\text{ kW} \cong \mathbf{2120\ kW}$$

The actual electric power generation is determined by multiplying the power generation potential by the efficiency,

$$\dot{W}_{\text{elect}} = \eta_{\text{wind turbine}}\dot{W}_{\max} = (0.30)(2121\text{ kW}) = \mathbf{636\ kW}$$

Therefore, 636 kW of actual power can be generated by this wind turbine at the stated conditions.




***Discussion*** The power generation of a wind turbine is proportional to the cube of the wind velocity, and thus the power generation will change strongly with the wind conditions.

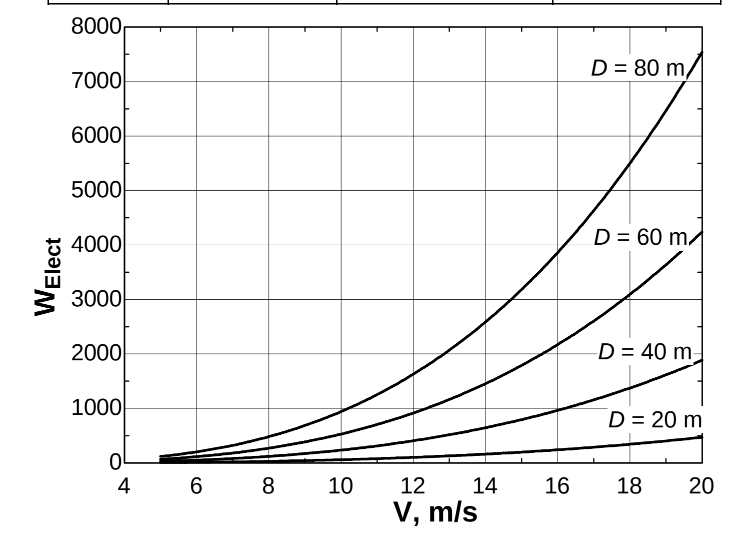
**5-22** 

**Solution** The previous problem is reconsidered. The effect of wind velocity and the blade span diameter on wind power generation as the velocity varies from 5 m/s to 20 m/s in increments of 5 m/s, and the diameter varies from 20 m to 80 m in increments of 20 m is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
D1=20 "m"
D2=40 "m"
D3=60 "m"
D4=80 "m"
Eta=0.30
rho=1.25 "kg/m3"
m1_dot=rho*V*(pi*D1^2/4); W1_Elect=Eta*m1_dot*(V^2/2)/1000 "kW"
m2_dot=rho*V*(pi*D2^2/4); W2_Elect=Eta*m2_dot*(V^2/2)/1000 "kW"
m3_dot=rho*V*(pi*D3^2/4); W3_Elect=Eta*m3_dot*(V^2/2)/1000 "kW"
m4_dot=rho*V*(pi*D4^2/4); W4_Elect=Eta*m4_dot*(V^2/2)/1000 "kW"
```

| $D$, m | $V$, m/s | $m$, kg/s | $W_{elect}$, kW |
|---|---|---|---|
| 20 | 5 | 1,963 | 7 |
| | 10 | 3,927 | 59 |
| | 15 | 5,890 | 199 |
| | 20 | 7,854 | 471 |
| 40 | 5 | 7,854 | 29 |
| | 10 | 15,708 | 236 |
| | 15 | 23,562 | 795 |
| | 20 | 31,416 | 1885 |
| 60 | 5 | 17,671 | 66 |
| | 10 | 35,343 | 530 |
| | 15 | 53,014 | 1789 |
| | 20 | 70,686 | 4241 |
| 80 | 5 | 31,416 | 118 |
| | 10 | 62,832 | 942 |
| | 15 | 94,248 | 3181 |
| | 20 | 125,664 | 7540 |




***Discussion*** Wind turbine power output is obviously nonlinear with respect to both velocity and diameter.

**5-23E**

**Solution** A differential thermocouple indicates that the temperature of water rises a certain amount as it flows through a pump at a specified rate. The mechanical efficiency of the pump is to be determined.

***Assumptions*** **1** The pump is adiabatic so that there is no heat transfer with the surroundings, and the temperature rise of water is completely due to frictional heating. **2** Water is an incompressible substance.

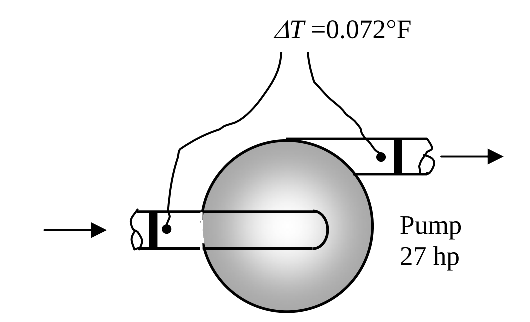
***Properties*** We take the density of water to be $\rho$ = 62.4 lbm/ft$^3$ and its specific heat to be $C$ = 1.0 Btu/lbm·°F.

***Analysis*** The increase in the temperature of water is due to the conversion of mechanical energy to thermal energy, and the amount of mechanical energy converted to thermal energy is equal to the increase in the internal energy of water,

$$\dot{m} = \rho \dot{V} = (62.4\ \text{lbm/ft}^3)(1.5\ \text{ft}^3/\text{s}) = 93.6\ \text{lbm/s}$$

$$\dot{E}_{\text{mech, loss}} = \Delta \dot{U} = \dot{m} c \Delta T$$

$$= (93.6\ \text{lbm/s})(1.0\ \text{Btu/lbm}\cdot{}^\circ\text{F})(0.072^\circ\text{F})\left(\frac{1\ \text{hp}}{0.7068\ \text{Btu/s}}\right) = 9.53\ \text{hp}$$




The mechanical efficiency of the pump is determined from the general definition of mechanical efficiency,

$$\eta_{\text{pump}} = 1 - \frac{\dot{E}_{\text{mech,loss}}}{\dot{W}_{\text{mech, in}}} = 1 - \frac{9.53\ \text{hp}}{27\ \text{hp}} = 0.647 \quad \text{or} \quad \mathbf{64.7\%}$$

***Discussion*** Note that despite the conversion of more than one-third of the mechanical power input into thermal energy, the temperature of water rises by only a small fraction of a degree. Therefore, the temperature rise of a fluid due to frictional heating is usually negligible in heat transfer analysis.

---

**5-24**

**Solution** Water is pumped from a lake to a storage tank at a specified rate. The overall efficiency of the pump-motor unit and the pressure difference between the inlet and the exit of the pump are to be determined.

***Assumptions*** **1** The elevations of the tank and the lake remain constant. **2** Frictional losses in the pipes are negligible. **3** The changes in kinetic energy are negligible. **4** The elevation difference across the pump is negligible.

***Properties*** We take the density of water to be $\rho$ = 1000 kg/m$^3$.

***Analysis*** (*a*) We take the free surface of the lake to be point 1 and the free surfaces of the storage tank to be point 2. We also take the lake surface as the reference level ($z_1$ = 0), and thus the potential energy at points 1 and 2 are $pe_1$ = 0 and $pe_2 = gz_2$. The flow energy at both points is zero since both 1 and 2 are open to the atmosphere ($P_1 = P_2 = P_{atm}$). Further, the kinetic energy at both points is zero ($ke_1 = ke_2 = 0$) since the water at both locations is essentially stationary. The mass flow rate of water and its potential energy at point 2 are

$$\dot{m} = \rho\dot{V} = (1000\ \text{kg/m}^3)(0.070\ \text{m}^3/\text{s}) = 70\ \text{kg/s}$$

$$pe_1 = gz_1 = (9.81\ \text{m/s}^2)(20\ \text{m})\left(\frac{1\ \text{kJ/kg}}{1000\ \text{m}^2/\text{s}^2}\right) = 0.196\ \text{kJ/kg}$$

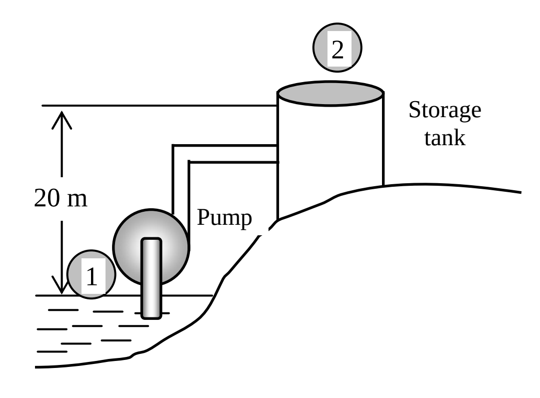
Then the rate of increase of the mechanical energy of water becomes

$$\Delta\dot{E}_{\text{mech,fluid}} = \dot{m}(e_{\text{mech,out}} - e_{\text{mech,in}}) = \dot{m}(pe_2 - 0) = \dot{m}pe_2 = (70\ \text{kg/s})(0.196\ \text{kJ/kg}) = 13.7\ \text{kW}$$

The overall efficiency of the combined pump-motor unit is determined from its definition,

$$\eta_{\text{pump-motor}} = \frac{\Delta\dot{E}_{\text{mech,fluid}}}{\dot{W}_{\text{elect,in}}} = \frac{13.7\ \text{kW}}{20.4\ \text{kW}} = 0.672 \quad \text{or} \quad \mathbf{67.2\%}$$

(*b*) Now we consider the pump. The change in the mechanical energy of water as it flows through the pump consists of the change in the flow energy only since the elevation difference across the pump and the change in the kinetic energy are negligible. Also, this change must be equal to the useful mechanical energy supplied by the pump, which is 13.7 kW:




$$\Delta\dot{E}_{\text{mech,fluid}} = \dot{m}(e_{\text{mech,out}} - e_{\text{mech,in}}) = \dot{m}\frac{P_2 - P_1}{\rho} = \dot{V}\Delta P$$

Solving for $\Delta P$ and substituting,

$$\Delta P = \frac{\Delta\dot{E}_{\text{mech,fluid}}}{\dot{V}} = \frac{13.7\ \text{kJ/s}}{0.070\ \text{m}^3/\text{s}}\left(\frac{1\ \text{kPa}\cdot\text{m}^3}{1\ \text{kJ}}\right) = \mathbf{196\ kPa}$$

Therefore, the pump must boost the pressure of water by 196 kPa in order to raise its elevation by 20 m.

***Discussion*** Note that only two-thirds of the electric energy consumed by the pump-motor is converted to the mechanical energy of water; the remaining one-third is wasted because of the inefficiencies of the pump and the motor.

## Bernoulli Equation

### 5-25C

**Solution** We are to define streamwise acceleration and discuss how it differs from normal acceleration.

***Analysis*** The **acceleration of a fluid particle along a streamline** is called *streamwise acceleration*, and it is due to a change in speed along a streamline. *Normal acceleration* (or centrifugal acceleration), on the other hand, is the **acceleration of a fluid particle in the direction normal to the streamline**, and it is due to a change in direction.

***Discussion*** In a general fluid flow problem, both streamwise and normal acceleration are present.

### 5-26C

**Solution** We are to express the Bernoulli equation in three different ways.

***Analysis*** The Bernoulli equation is expressed in three different ways as follows:

*(a)* In terms of energies: $\frac{P}{\rho}+\frac{V^2}{2}+gz=\text{constant}$

*(b)* In terms of pressures: $P+\rho\frac{V^2}{2}+\rho gz=\text{constant}$

*(c)* in terms of heads: $\frac{P}{\rho g}+\frac{V^2}{2g}+z=H=\text{constant}$

***Discussion*** You could, of course, express it in other ways, but these three are the most useful.

### 5-27C

**Solution** We are to discuss the three major assumptions used in the derivation of the Bernoulli equation.

***Analysis*** The three major assumptions used in the derivation of the Bernoulli equation are that the flow is **steady**, **there is negligible frictional effects**, and the flow is **incompressible**.

***Discussion*** If any one of these assumptions is not valid, the Bernoulli equation should not be used. Unfortunately, many people use it anyway, leading to errors.

### 5-28C

**Solution** We are to define and discuss static, dynamic, and hydrostatic pressure.

***Analysis*** *Static pressure* $P$ is the **actual pressure of the fluid**. *Dynamic pressure* $\rho V^2/2$ is the **pressure rise when the fluid in motion is brought to a stop isentropically**. *Hydrostatic pressure* $\rho gz$ is not pressure in a real sense since its value depends on the reference level selected, and it **accounts for the effects of fluid weight on pressure**. The sum of static, dynamic, and hydrostatic pressures is constant when flow is steady and incompressible, and when frictional effects are negligible.

***Discussion*** The incompressible Bernoulli equation states that the sum of these three pressures is constant along a streamline; this approximation is valid only for steady and incompressible flow with negligible frictional effects.

**5-29C**

**Solution** We are to define and discuss pressure head, velocity head, and elevation head.

***Analysis*** The **sum of the static and dynamic pressures** is called the *stagnation pressure*, and it is expressed as $\boxed{P_{\text{stag}} = P + \rho V^2 / 2}$. The stagnation pressure can be measured by **a Pitot tube whose inlet is normal to the flow**.

***Discussion*** Stagnation pressure, as its name implies, is the pressure obtained when a flowing fluid is brought to rest isentropically, at a so-called *stagnation point*.

---

**5-30C**

**Solution** We are to define and discuss pressure head, velocity head, and elevation head.

***Analysis*** The *pressure head* $P/\rho g$ is the **height of a fluid column that produces the static pressure *P***. The *velocity head* $V^2/2$ is the **elevation needed for a fluid to reach the velocity *V* during frictionless free fall**. The *elevation head* $z$ is the **height of a fluid relative to a reference level**.

***Discussion*** It is often convenient in fluid mechanics to work with *head* – pressure expressed as an equivalent column height of fluid.

---

**5-31C**

**Solution** We are to define hydraulic grade line and compare it to energy grade line.

***Analysis*** **The curve that represents the sum of the static pressure and the elevation heads**, $P/\rho g + z$, is called the *hydraulic grade line* or HGL. The curve that represents the total head of the fluid, $P/\rho g + V^2/2g + z$, is called the *energy line* or EGL. Thus, in comparison, the energy grade line contains an extra kinetic-energy-type term. **For stationary bodies such as reservoirs or lakes, the EL and HGL coincide with the free surface of the liquid**.

***Discussion*** The hydraulic grade line can rise or fall along flow in a pipe or duct as the cross-sectional area increases or decreases, whereas the energy grade line *always* decreases unless energy is added to the fluid (like with a pump).

---

**5-32C**

**Solution** We are to discuss the hydraulic grade line in open-channel flow and at the outlet of a pipe.

***Analysis*** For *open-channel flow*, the hydraulic grade line (HGL) **coincides with the free surface of the liquid**. At the exit of a pipe discharging to the atmosphere, HGL **coincides with the elevation of the pipe outlet**.

***Discussion*** We are assuming incompressible flow, and the pressure at the pipe outlet is atmospheric.

---

**5-33C**

**Solution** We are to discuss the maximum rise of a jet of water from a tank.

***Analysis*** With no losses and a 100% efficient nozzle, the water stream could reach to the water level in the tank, or 20 meters. In reality, friction losses in the hose, nozzle inefficiencies, orifice losses, and air drag would prevent attainment of the maximum theoretical height.

***Discussion*** In fact, the actual maximum obtainable height is much smaller than this ideal theoretical limit.

---

**5-34C**

**Solution** We are to discuss the effect of liquid density on the operation of a siphon.

***Analysis*** The lower density liquid can go over a higher wall, provided that cavitation pressure is not reached. Therefore, oil may be able to go over a higher wall than water.

***Discussion*** However, frictional losses in the flow of oil in a pipe or tube are much greater than those of water since the viscosity of oil is much greater than that of water. When frictional losses are considered, the water may actually be able to be siphoned over a higher wall than the oil, depending on the tube diameter and length, etc.

---

**5-35C**

**Solution** We are to explain how and why a siphon works, and its limitations.

***Analysis*** Siphoning works because of the elevation and thus pressure difference between the inlet and exit of a tube. The pressure at the tube exit and at the free surface of a liquid is the atmospheric pressure. When the tube exit is below the free surface of the liquid, the elevation head difference drives the flow through the tube. At sea level, 1 atm pressure can support about 10.3 m of cold water (cold water has a low vapor pressure). Therefore, siphoning cold water over a 7 m wall is **theoretically feasible**.

***Discussion*** In actual practice, siphoning is also limited by frictional effects in the tube, and by cavitation.

---

**5-36C**

**Solution** We are to compare siphoning at sea level and on a mountain.

***Analysis*** At sea level, a person can theoretically siphon water over a wall as high as 10.3 m. At the top of a high mountain where the pressure is about half of the atmospheric pressure at sea level, a person can theoretically siphon water over a wall that is only half as high. **An atmospheric pressure of 58.5 kPa is insufficient to support a 8.5 meter high siphon.**

***Discussion*** In actual practice, siphoning is also limited by frictional effects in the tube, and by cavitation.

---

**5-37C**

**Solution** We are to analyze the pressure change in a converging duct.

***Analysis*** As the duct converges to a smaller cross-sectional area, the velocity increases. By Bernoulli's equation, the pressure therefore decreases. Thus **Manometer A** is correct since the pressure on the right side of the manometer is obviously smaller. According to the Bernoulli approximation, the fluid levels in the manometer are independent of the flow direction, and reversing the flow direction would have no effect on the manometer levels. **Manometer A is still correct if the flow is reversed**.

***Discussion*** In reality, it is hard for a fluid to expand without the flow separating from the walls. Thus, reverse flow with such a sharp expansion would not produce as much of a pressure rise as that predicted by the Bernoulli approximation.

---

**5-38C**

**Solution** We are to discuss and compare two different types of manometer arrangements in a flow.

***Analysis*** Arrangement 1 consists of a Pitot probe that measures the stagnation pressure at the pipe centerline, along with a static pressure tap that measures static pressure at the bottom of the pipe. Arrangement 2 is a Pitot-static probe that measures both stagnation pressure and static pressure at *nearly the same location* at the pipe centerline. Because of this, **arrangement 2 is more accurate**. However, it turns out that static pressure in a pipe varies with elevation across the pipe cross section in much the same way as in hydrostatics. Therefore, arrangement 1 is also very accurate, and the elevation difference between the Pitot probe and the static pressure tap is nearly compensated by the change in hydrostatic pressure. Since elevation changes are not important in either arrangement, there is **no change in our analysis when the water is replaced by air**.

***Discussion*** Ignoring the effects of gravity, the pressure at the centerline of a turbulent pipe flow is actually somewhat smaller than that at the wall due to the turbulent eddies in the flow, but this effect is small.

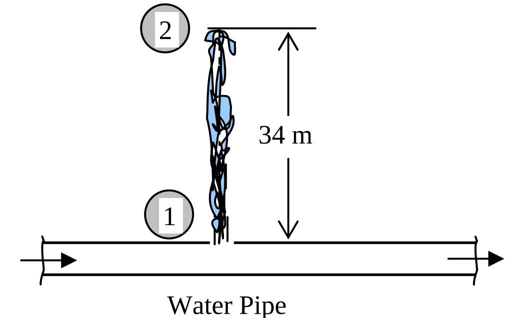
---

**5-39**

**Solution** A water pipe bursts as a result of freezing, and water shoots up into the air a certain height. The gage pressure of water in the pipe is to be determined.

***Assumptions*** **1** The flow is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** The water pressure in the pipe at the burst section is equal to the water main pressure. **3** Friction between the water and air is negligible. **4** The irreversibilities that may occur at the burst section of the pipe due to abrupt expansion are negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$.




***Analysis*** This problem involves the conversion of flow, kinetic, and potential energies to each other without involving any pumps, turbines, and wasteful components with large frictional losses, and thus it is suitable for the use of the Bernoulli equation. The water height will be maximum under the stated assumptions. The velocity inside the hose is relatively low ($V_1 \cong 0$) and we take the burst section of the pipe as the reference level ($z_1 = 0$). At the top of the water trajectory $V_2 = 0$, and atmospheric pressure pertains. Then the Bernoulli equation simplifies to

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \quad \rightarrow \quad \frac{P_1}{\rho g} = \frac{P_{atm}}{\rho g} + z_2 \quad \rightarrow \quad \frac{P_1 - P_{atm}}{\rho g} = z_2 \quad \rightarrow \quad \frac{P_{1,\text{gage}}}{\rho g} = z_2$$

Solving for $P_{1,\text{gage}}$ and substituting,

$$P_{1,\text{gage}} = \rho g z_2 = (1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(34\ \text{m})\left(\frac{1\ \text{kPa}}{1\ \text{kN/m}^2}\right)\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) = \mathbf{334\ kPa}$$

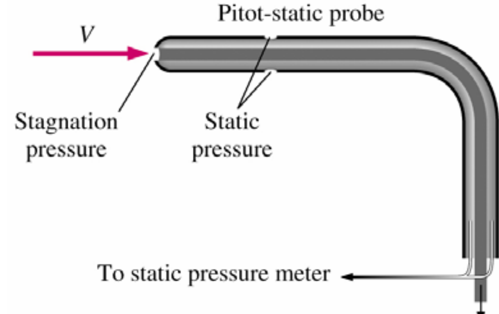
Therefore, the pressure in the main must be at least 334 kPa above the atmospheric pressure.

***Discussion*** The result obtained by the Bernoulli equation represents a limit, since frictional losses are neglected, and should be interpreted accordingly. It tells us that the water pressure (gage) cannot possibly be less than 334 kPa (giving us a lower limit), and in all likelihood, the pressure will be much higher.

---

**5-40**

**Solution** The velocity of an aircraft is to be measured by a Pitot-static probe. For a given differential pressure reading, the velocity of the aircraft is to be determined.




***Assumptions*** **1** The air flow over the aircraft is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** Standard atmospheric conditions exist. **3** The wind effects are negligible.

***Properties*** The density of the atmosphere at an elevation of 3000 m is $\rho$ = 0.909 kg/m$^3$.

***Analysis*** We take point 1 at the entrance of the tube whose opening is parallel to flow, and point 2 at the entrance of the tube whose entrance is normal to flow. Noting that point 2 is a stagnation point and thus $V_2$ = 0 and $z_1 = z_2$, the application of the Bernoulli equation between points 1 and 2 gives

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad \frac{V_1^2}{2g}=\frac{P_2-P_1}{\rho g} \quad\rightarrow\quad \frac{V_1^2}{2}=\frac{P_{stag}-P_1}{\rho}$$

Solving for $V_1$ and substituting,

$$V_1=\sqrt{\frac{2(P_{stag}-P_1)}{\rho}}=\sqrt{\frac{2(3000\text{ N/m}^2)}{0.909\text{ kg/m}^3}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)}=\mathbf{81.2\ m/s}=\mathbf{292\ km/h}$$

since 1 Pa = 1 N/m$^2$ and 1 m/s = 3.6 km/h.

***Discussion*** Note that the velocity of an aircraft can be determined by simply measuring the differential pressure on a Pitot-static probe.

---

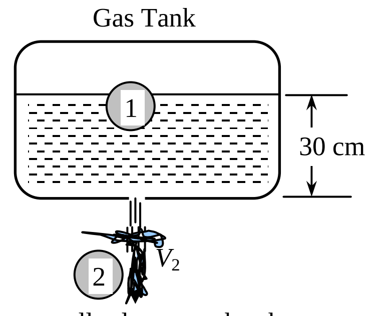
**5-41**

**Solution** The bottom of a car hits a sharp rock and a small hole develops at the bottom of its gas tank. For a given height of gasoline, the initial velocity of the gasoline out of the hole is to be determined. Also, the variation of velocity with time and the effect of the tightness of the lid on flow rate are to be discussed.

***Assumptions*** **1** The flow is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** The air space in the tank is at atmospheric pressure. **3** The splashing of the gasoline in the tank during travel is not considered.

***Analysis*** This problem involves the conversion of flow, kinetic, and potential energies to each other without involving any pumps, turbines, and wasteful components with large frictional losses, and thus it is suitable for the use of the Bernoulli equation. We take point 1 to be at the free surface of gasoline in the tank so that $P_1 = P_{atm}$ (open to the atmosphere) $V_1 \cong 0$ (the tank is large relative to the outlet), and $z_1$ = 0.3 m and $z_2$ = 0 (we take the reference level at the hole. Also, $P_2 = P_{atm}$ (gasoline discharges into the atmosphere). Then the Bernoulli equation simplifies to

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad z_1=\frac{V_2^2}{2g}$$




Solving for $V_2$ and substituting,

$$V_2=\sqrt{2gz_1}=\sqrt{2(9.81\text{ m/s}^2)(0.3\text{ m})}=\mathbf{2.43\ m/s}$$

Therefore, the gasoline will initially leave the tank with a velocity of 2.43 m/s.

***Discussion*** The Bernoulli equation applies along a streamline, and streamlines generally do not make sharp turns. The velocity will be less than 2.43 m/s since the hole is probably sharp-edged and it will cause some head loss. As the gasoline level is reduced, the velocity will decrease since velocity is proportional to the square root of liquid height. If the lid is tightly closed and no air can replace the lost gasoline volume, the pressure above the gasoline level will be reduced, and the velocity will be decreased.

---

**5-42E** **[*Also solved using EES on enclosed DVD*]**

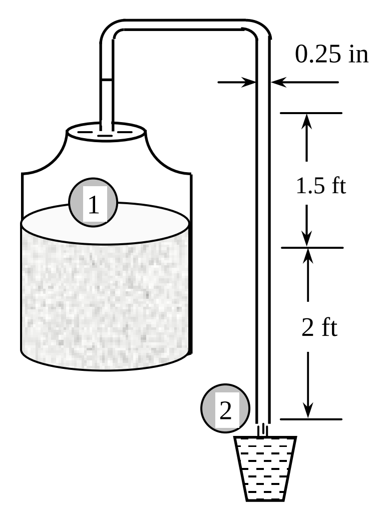
**Solution** The drinking water needs of an office are met by large water bottles with a plastic hose inserted in it. The minimum filling time of an 8-oz glass is to be determined when the bottle is full and when it is near empty.

***Assumptions*** **1** The flow is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** All losses are neglected to obtain the minimum filling time.

***Analysis*** We take point 1 to be at the free surface of water in the bottle and point 2 at the exit of the tube so that $P_1 = P_2 = P_{atm}$ (the bottle is open to the atmosphere and water discharges into the atmosphere), $V_1 \cong 0$ (the bottle is large relative to the tube diameter), and $z_2 = 0$ (we take point 2 as the reference level). Then the Bernoulli equation simplifies to

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad z_1=\frac{V_2^2}{2g} \quad\rightarrow\quad V_2=\sqrt{2gz_1}$$

Substituting, the discharge velocity of water and the filling time are determined as follows:




(*a*) *Full bottle* ($z_1$ = 3.5 ft):

$$V_2=\sqrt{2(32.2\text{ ft/s}^2)(3.5\text{ ft})}=15.0\text{ ft/s}$$

$$A=\pi D^2/4=\pi(0.25/12\text{ ft})^2/4=3.41\times10^{-4}\text{ ft}^2$$

$$\Delta t=\frac{V}{\dot V}=\frac{V}{AV_2}=\frac{0.00835\text{ ft}^3}{(3.41\times10^{-4}\text{ ft}^2)(15\text{ ft/s})}=\mathbf{1.6\ s}$$

(*b*) *Empty bottle* ($z_1$ = 2 ft):

$$V_2=\sqrt{2(32.2\text{ ft/s}^2)(2\text{ ft})}=11.3\text{ ft/s}$$

$$\Delta t=\frac{V}{\dot V}=\frac{V}{AV_2}=\frac{0.00835\text{ ft}^3}{(3.41\times10^{-4}\text{ ft}^2)(11.3\text{ ft/s})}=\mathbf{2.2\ s}$$

***Discussion*** The siphoning time is determined assuming frictionless flow, and thus this is the *minimum time* required. In reality, the time will be longer because of friction between water and the tube surface.

---

**5-43**

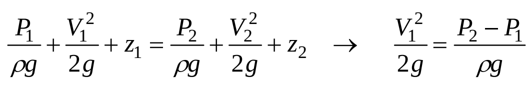
**Solution** The static and stagnation pressures in a horizontal pipe are measured. The velocity at the center of the pipe is to be determined.

***Assumptions*** The flow is steady, incompressible, and irrotational with negligible frictional effects in the short distance between the two pressure measurement locations (so that the Bernoulli equation is applicable).

***Analysis*** We take points 1 and 2 along the centerline of the pipe, with point 1 directly under the piezometer and point 2 at the entrance of the Pitot-static probe (the stagnation point).

This is a steady flow with straight and parallel streamlines, and thus the static pressure at any point is equal to the hydrostatic pressure at that point. Noting that point 2 is a stagnation point and thus $V_2 = 0$ and $z_1 = z_2$, the application of the Bernoulli equation between points 1 and 2 gives

20 cm
35 cm
Water
1
2
V



$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad \frac{V_1^2}{2g}=\frac{P_2-P_1}{\rho g}$$

Substituting the $P_1$ and $P_2$ expressions give

$$\frac{V_1^2}{2g}=\frac{P_2-P_1}{\rho g}=\frac{\rho g(h_{pitot}+R)-\rho g(h_{piezo}+R)}{\rho g}=\frac{\rho g(h_{pitot}-h_{piezo})}{\rho g}=h_{pitot}-h_{piezo}$$

Solving for $V_1$ and substituting,

$$V_1=\sqrt{2g(h_{pitot}-h_{piezo})}=\sqrt{2(9.81\text{ m/s}^2)[(0.35-0.20)\text{ m}]}=\mathbf{1.72\ m/s}$$

***Discussion*** Note that to determine the flow velocity, all we need is to measure the height of the excess fluid column in the Pitot-static probe.

---

**5-44**

**Solution** A water tank of diameter $D_o$ and height $H$ open to the atmosphere is initially filled with water. An orifice of diameter $D$ with a smooth entrance (no losses) at the bottom drains to the atmosphere. Relations are to be developed for the time required for the tank to empty completely and half-way.

***Assumptions*** **1** The orifice has a smooth entrance, and thus the frictional losses are negligible. **2** The flow is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable).

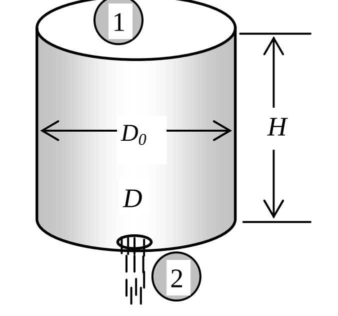
***Analysis*** We take point 1 at the free surface of the tank, and point 2 at the exit of orifice. We take the reference level at the orifice ($z_2 = 0$), and take the positive direction of $z$ to be upwards. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the Bernoulli equation between these two points simplifies to

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad z_1=\frac{V_2^2}{2g} \quad\rightarrow\quad V_2=\sqrt{2gz_1}$$

For generality, we express the water height in the tank at any time $t$ by $z$, and the discharge velocity by $V_2 = \sqrt{2gz}$. Note that water surface in the tank moves down as the tank drains, and thus $z$ is a variable whose value changes from $H$ at the beginning to 0 when the tank is emptied completely.

We denote the diameter of the orifice by $D$, and the diameter of the tank by $D_o$. The flow rate of water from the tank is obtained by multiplying the discharge velocity by the orifice cross-sectional area,

$$\dot{V} = A_{\text{orifice}}V_2 = \frac{\pi D^2}{4}\sqrt{2gz}$$




Then the amount of water that flows through the orifice during a differential time interval $dt$ is

$$dV = \dot{V}dt = \frac{\pi D^2}{4}\sqrt{2gz}\,dt \qquad (1)$$

which, from conservation of mass, must be equal to the decrease in the volume of water in the tank,

$$dV = A_{\text{tank}}(-dz) = -\frac{\pi D_0^2}{4}dz \qquad (2)$$

where $dz$ is the change in the water level in the tank during $dt$. (Note that $dz$ is a negative quantity since the positive direction of $z$ is upwards. Therefore, we used $-dz$ to get a positive quantity for the amount of water discharged). Setting Eqs. (1) and (2) equal to each other and rearranging,

$$\frac{\pi D^2}{4}\sqrt{2gz}\,dt = -\frac{\pi D_0^2}{4}dz \quad\rightarrow\quad dt = -\frac{D_0^2}{D^2}\sqrt{\frac{1}{2gz}}dz = -\frac{D_0^2}{D^2\sqrt{2g}}z^{-\frac{1}{2}}dz$$

The last relation can be integrated easily since the variables are separated. Letting $t_f$ be the discharge time and integrating it from $t = 0$ when $z = z_i = H$ to $t = t_f$ when $z = z_f$ gives

$$\int_{t=0}^{t_f}dt = -\frac{D_0^2}{D^2\sqrt{2g}}\int_{z=z_1}^{z_f}z^{-1/2}dz \quad\rightarrow\quad t_f = -\frac{D_0^2}{D^2\sqrt{2g}}\left.\frac{z^{\frac{1}{2}}}{\frac{1}{2}}\right|_{z_1}^{z_f} = \frac{2D_0^2}{D^2\sqrt{2g}}\left(\sqrt{z_i}-\sqrt{z_f}\right) = \frac{D_0^2}{D^2}\left(\sqrt{\frac{2z_i}{g}}-\sqrt{\frac{2z_f}{g}}\right)$$

Then the discharging time for the two cases becomes as follows:

(*a*) *The tank empties halfway*: $z_i = H$ and $z_f = H/2$: $\boxed{t_f = \frac{D_0^2}{D^2}\left(\sqrt{\frac{2H}{g}}-\sqrt{\frac{H}{g}}\right)}$

(*b*) *The tank empties completely*: $z_i = H$ and $z_f = 0$: $\boxed{t_f = \frac{D_0^2}{D^2}\sqrt{\frac{2H}{g}}}$

***Discussion*** Note that the discharging time is inversely proportional to the square of the orifice diameter. Therefore, the discharging time can be reduced to one-fourth by doubling the diameter of the orifice.

**5-45**

**Solution** Water discharges to the atmosphere from the orifice at the bottom of a pressurized tank. Assuming frictionless flow, the discharge rate of water from the tank is to be determined.

***Assumptions*** **1** The orifice has a smooth entrance, and thus the frictional losses are negligible. **2** The flow is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable).

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** We take point 1 at the free surface of the tank, and point 2 at the exit of orifice, which is also taken to be the reference level ($z_2$ = 0). Noting that the fluid velocity at the free surface is very low ($V_1 \cong 0$) and water discharges into the atmosphere (and thus $P_2 = P_{atm}$), the Bernoulli equation simplifies to

1
Air, 300 kPa
Water Tank
3 m
2
10 cm

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad \rightarrow \quad \frac{V_2^2}{2g}=\frac{P_1-P_2}{\rho g}+z_1$$

Solving for $V_2$ and substituting, the discharge velocity is determined to

$$V_2=\sqrt{\frac{2(P_1-P_2)}{\rho}+2gz_1}=\sqrt{\frac{2(300-100)\text{ kPa}}{1000\text{ kg/m}^3}\left(\frac{1000\text{ N/m}^2}{1\text{ kPa}}\right)\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)+2(9.81\text{ m/s}^2)(3\text{ m})}$$
$$=21.4\text{ m/s}$$

Then the initial rate of discharge of water becomes

$$\dot{V}=A_{\text{orifice}}V_2=\frac{\pi D^2}{4}V_2=\frac{\pi(0.10\text{ m})^2}{4}(21.4\text{ m/s})=\mathbf{0.168\ m^3/s}$$

***Discussion*** Note that this is the maximum flow rate since the frictional effects are ignored. Also, the velocity and the flow rate will decrease as the water level in the tank decreases.


**5-46** 

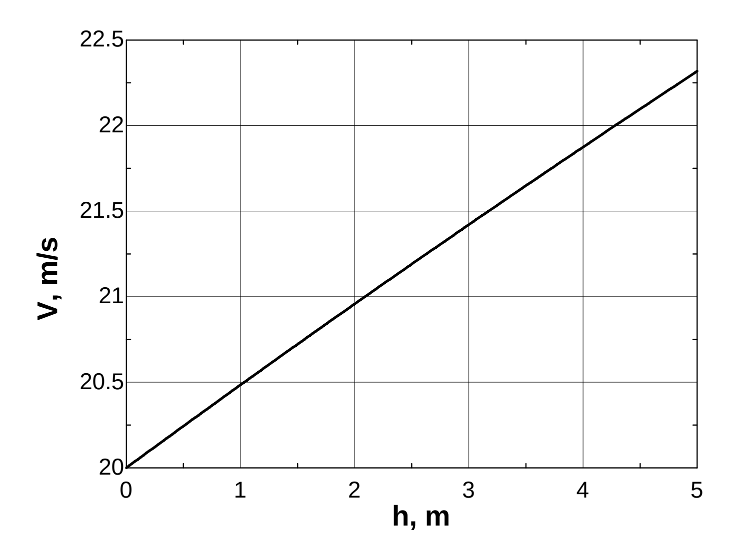
**Solution** The previous problem is reconsidered. The effect of water height in the tank on the discharge velocity as the water height varies from 0 to 5 m in increments of 0.5 m is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
g=9.81 "m/s2"
rho=1000 "kg/m3"
d=0.10 "m"

P1=300 "kPa"
P_atm=100 "kPa"
V=SQRT(2*(P1-P_atm)*1000/rho+2*g*h)
Ac=pi*D^2/4
V_dot=Ac*V
```

| *h*, m | *V*, m/s | $\dot{V}$, m³/s |
|---|---|---|
| 0.00 | 20.0 | 0.157 |
| 0.50 | 20.2 | 0.159 |
| 1.00 | 20.5 | 0.161 |
| 1.50 | 20.7 | 0.163 |
| 2.00 | 21.0 | 0.165 |
| 2.50 | 21.2 | 0.166 |
| 3.00 | 21.4 | 0.168 |
| 3.50 | 21.6 | 0.170 |
| 4.00 | 21.9 | 0.172 |
| 4.50 | 22.1 | 0.174 |
| 5.00 | 22.3 | 0.175 |

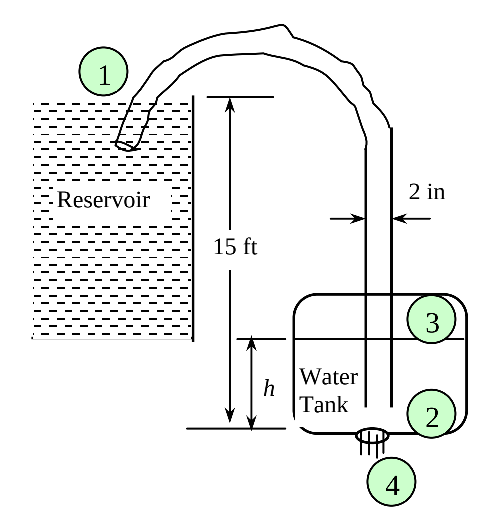



***Discussion*** Velocity appears to change nearly linearly with *h* in this range of data, but the relationship is *not* linear.

**5-47E**

**Solution** A siphon pumps water from a large reservoir to a lower tank which is initially empty. Water leaves the tank through an orifice. The height the water will rise in the tank at equilibrium is to be determined.

***Assumptions*** **1** The flow is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** Both the tank and the reservoir are open to the atmosphere. **3** The water level of the reservoir remains constant.




***Analysis*** We take the reference level to be at the bottom of the tank, and the water height in the tank at any time to be $h$. We take point 1 to be at the free surface of reservoir, point 2 at the exit of the siphon, which is placed at the bottom of the tank, and point 3 at the free surface of the tank, and point 4 at the exit of the orifice at the bottom of the tank. Then $z_1$ = 20 ft, $z_2 = z_4 = 0$, $z_3 = h$, $P_1 = P_3 = P_4 = P_{atm}$ (the reservoir is open to the atmosphere and water discharges into the atmosphere) $P_2 = P_{atm}+\rho gh$ (the hydrostatic pressure at the bottom of the tank where the siphon discharges), and $V_1 \cong V_3 \cong 0$ (the free surfaces of reservoir and the tank are large relative to the tube diameter). Then the Bernoulli equation between 1-2 and 3-4 simplifies to

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\to\quad \frac{P_{atm}}{\rho g}+z_1=\frac{P_{atm}+\rho gh}{\rho g}+\frac{V_2^2}{2g} \quad\to\quad V_2=\sqrt{2gz_1-2gh}=\sqrt{2g(z_1-h)}$$

$$\frac{P_3}{\rho g}+\frac{V_3^2}{2g}+z_3=\frac{P_4}{\rho g}+\frac{V_4^2}{2g}+z_4 \quad\to\quad h=\frac{V_4^2}{2g} \quad\to\quad V_4=\sqrt{2gh}$$

Noting that the diameters of the tube and the orifice are the same, the flow rates of water into and out of the tank will be the same when the water velocities in the tube and the orifice are equal since

$$\dot V_2=\dot V_4 \quad\to\quad AV_2=AV_4 \quad\to\quad V_2=V_4$$

Setting the two velocities equal to each other gives

$$V_2=V_4 \quad\to\quad \sqrt{2g(z_1-h)}=\sqrt{2gh} \quad\to\quad z_1-h=h \quad\to\quad h=\frac{z_1}{2}=\frac{15\text{ ft}}{2}=\mathbf{7.5\ ft}$$

Therefore, the water level in the tank will stabilize when the water level rises to 7.5 ft.

***Discussion*** This result is obtained assuming negligible friction. The result would be somewhat different if the friction in the pipe and orifice were considered.

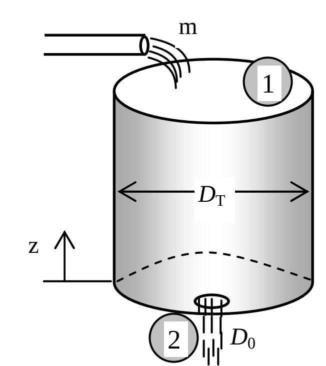
**5-48**

**Solution** Water enters an empty tank steadily at a specified rate. An orifice at the bottom allows water to escape. The maximum water level in the tank is to be determined, and a relation for water height $z$ as a function of time is to be obtained.

***Assumptions*** **1** The orifice has a smooth entrance, and thus the frictional losses are negligible. **2** The flow through the orifice is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable).

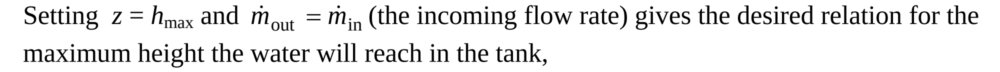
***Analysis*** (*a*) We take point 1 at the free surface of the tank, and point 2 at the exit of orifice. We take the reference level at the orifice ($z_2 = 0$), and take the positive direction of $z$ to be upwards. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$) (it becomes zero when the water in the tank reaches its maximum level), the Bernoulli equation between these two points simplifies to

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad z_1=\frac{V_2^2}{2g} \quad\rightarrow\quad V_2=\sqrt{2gz_1}$$




Then the mass flow rate through the orifice for a water height of $z$ becomes

$$\dot m_{out}=\rho\dot V_{out}=\rho A_{orifice}V_2=\rho\frac{\pi D_0^2}{4}\sqrt{2gz} \quad\rightarrow\quad z=\frac{1}{2g}\left(\frac{4\dot m_{out}}{\rho\pi D_0^2}\right)^2$$



Setting $z = h_{max}$ and $\dot m_{out} = \dot m_{in}$ (the incoming flow rate) gives the desired relation for the maximum height the water will reach in the tank,

$$h_{max}=\frac{1}{2g}\left(\frac{4\dot m_{in}}{\rho\pi D_0^2}\right)^2$$

(*b*) The amount of water that flows through the orifice and the increase in the amount of water in the tank during a differential time interval $dt$ are

$$dm_{out}=\dot m_{out}dt=\rho\frac{\pi D_0^2}{4}\sqrt{2gz}\,dt$$

$$dm_{tank}=\rho A_{tank}dz=\rho\frac{\pi D_T^2}{4}dz$$

The amount of water that enters the tank during $dt$ is $dm_{in}=\dot m_{in}dt$ (Recall that $\dot m_{in}$ = constant). Substituting them into the conservation of mass relation $dm_{tank}=dm_{in}-dm_{out}$ gives

$$dm_{tank}=\dot m_{in}dt-\dot m_{out}dt \quad\rightarrow\quad \rho\frac{\pi D_T^2}{4}dz=\left(\dot m_{in}-\rho\frac{\pi D_0^2}{4}\sqrt{2gz}\right)dt$$

Separating the variables, and integrating it from $z = 0$ at $t = 0$ to $z = z$ at time $t = t$ gives

$$\frac{\frac{1}{4}\rho\pi D_T^2dz}{\dot m_{in}-\frac{1}{4}\rho\pi D_0^2\sqrt{2gz}}=dt \quad\rightarrow\quad \int_{z=0}^{z}\frac{\frac{1}{4}\rho\pi D_T^2dz}{\dot m_{in}-\frac{1}{4}\rho\pi D_0^2\sqrt{2gz}}=\int_{t=0}^{t}dt=t$$

Performing the integration, the desired relation between the water height $z$ and time $t$ is obtained to be

$$\boxed{\frac{\frac{1}{2}\rho\pi D_T^2}{(\frac{1}{4}\rho\pi D_0^2\sqrt{2g})^2}\left(\frac{1}{4}\rho\pi D_0^2\sqrt{2gz}-\dot m_{in}\ln\frac{\dot m_{in}-\frac{1}{4}\rho\pi D_0^2\sqrt{2gz}}{\dot m_{in}}\right)=t}$$

***Discussion*** Note that this relation is implicit in $z$, and thus we can't obtain a relation in the form $z = f(t)$. Substituting a $z$ value in the left side gives the time it takes for the fluid level in the tank to reach that level. Equation solvers such as EES can easily solve implicit equations like this.

**5-49E**

**Solution** Water flows through a horizontal pipe that consists of two sections at a specified rate. The differential height of a mercury manometer placed between the two pipe sections is to be determined.

***Assumptions*** **1**The flow through the pipe is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** The losses in the reducing section are negligible.

***Properties*** The densities of mercury and water are $\rho_{Hg}$ = 847 lbm/ft$^3$ and $\rho_w$ = 62.4 lbm/ft$^3$.

***Analysis*** We take points 1 and 2 along the centerline of the pipe over the two tubes of the manometer. Noting that $z_1 = z_2$, the Bernoulli equation between points 1 and 2 gives

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad P_1-P_2=\frac{\rho_w(V_2^2-V_1^2)}{2} \qquad (1)$$

We let the differential height of the mercury manometer be $h$ and the distance between the centerline and the mercury level in the tube where mercury is raised be $s$. Then the pressure difference $P_2 - P_1$ can also be expressed as

$$P_1+\rho_w g(s+h)=P_2+\rho_w gs+\rho_{Hg}gh \quad\rightarrow\quad P_1-P_2=(\rho_{Hg}-\rho_w)gh \qquad (2)$$

Combining Eqs. (1) and (2) and solving for $h$,

$$\frac{\rho_w(V_2^2-V_1^2)}{2}=(\rho_{Hg}-\rho_w)gh \quad\rightarrow\quad h=\frac{\rho_w(V_2^2-V_1^2)}{2g(\rho_{Hg}-\rho_w)}=\frac{V_2^2-V_1^2}{2g(\rho_{Hg}/\rho_w-1)}$$

Calculating the velocities and substituting,

$$V_1=\frac{\dot V}{A_1}=\frac{\dot V}{\pi D_1^2/4}=\frac{1\text{ gal/s}}{\pi(4/12\text{ ft})^2/4}\left(\frac{0.13368\text{ ft}^3}{1\text{ gal}}\right)=1.53\text{ ft/s}$$

$$V_2=\frac{\dot V}{A_2}=\frac{\dot V}{\pi D_2^2/4}=\frac{1\text{ gal/s}}{\pi(2/12\text{ ft})^2/4}\left(\frac{0.13368\text{ ft}^3}{1\text{ gal}}\right)=6.13\text{ ft/s}$$

$$h=\frac{(6.13\text{ ft/s})^2-(1.53\text{ ft/s})^2}{2(32.2\text{ ft/s}^2)(847/62.4-1)}=0.0435\text{ ft}=\mathbf{0.52\text{ in}}$$

Therefore, the differential height of the mercury column will be 0.52 in.

***Discussion*** In reality, there are frictional losses in the pipe, and the pressure at location 2 will actually be smaller than that estimated here, and therefore $h$ will be larger than that calculated here.

**5-50**

**Solution** An airplane is flying at a certain altitude at a given speed. The pressure on the stagnation point on the nose of the plane is to be determined, and the approach to be used at high velocities is to be discussed.

***Assumptions*** **1** The air flow over the aircraft is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** Standard atmospheric conditions exist. **3** The wind effects are negligible.

***Properties*** The density of the atmospheric air at an elevation of 12,000 m is $\rho$ = 0.312 kg/m$^3$.

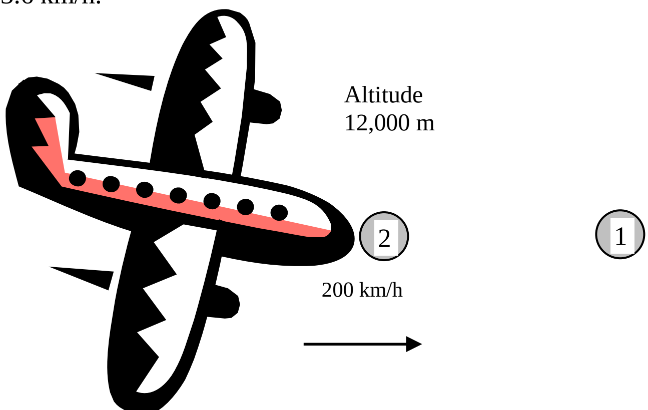
***Analysis*** We take point 1 well ahead of the plane at the level of the nose, and point 2 at the nose where the flow comes to a stop. Noting that point 2 is a stagnation point and thus $V_2$ = 0 and $z_1$ = $z_2$, the application of the Bernoulli equation between points 1 and 2 gives

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad \frac{V_1^2}{2g}=\frac{P_2-P_1}{\rho g} \quad\rightarrow\quad \frac{V_1^2}{2}=\frac{P_{stag}-P_{\text{atm}}}{\rho}=\frac{P_{\text{stag, gage}}}{\rho}$$

Solving for $P_{\text{stag, gage}}$ and substituting,

$$P_{\text{stag, gage}}=\frac{\rho V_1^2}{2}=\frac{(0.312\text{ kg/m}^3)(200/3.6\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)=481\text{ N/m}^2=\mathbf{481\ Pa}$$

since 1 Pa = 1 N/m$^2$ and 1 m/s = 3.6 km/h.




***Discussion*** A flight velocity of 1050 km/h = 292 m/s corresponds to a Mach number much greater than 0.3 (the speed of sound is about 340 m/s at room conditions, and lower at higher altitudes, and thus a Mach number of 292/340 = 0.86). Therefore, the flow can no longer be assumed to be incompressible, and the Bernoulli equation given above cannot be used. This problem can be solved using the modified Bernoulli equation that accounts for the effects of compressibility, assuming isentropic flow.

**5-51**

**Solution** A Pitot-static probe is inserted into the duct of an air heating system parallel to flow, and the differential height of the water column is measured. The flow velocity and the pressure rise at the tip of the Pitot-static probe are to be determined.

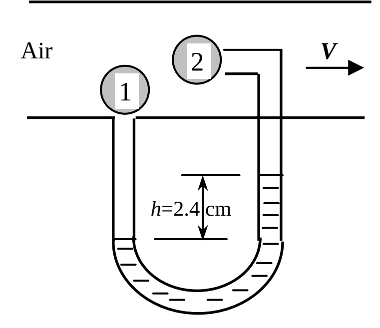
***Assumptions*** **1** The flow through the duct is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** Air is an ideal gas.

***Properties*** We take the density of water to be $\rho$ = 1000 kg/m$^3$. The gas constant of air is $R$ = 0.287 kPa·m$^3$/kg·K.

***Analysis*** We take point 1 on the side of the probe where the entrance is parallel to flow and is connected to the static arm of the Pitot-static probe, and point 2 at the tip of the probe where the entrance is normal to flow and is connected to the dynamic arm of the Pitot-static probe. Noting that point 2 is a stagnation point and thus $V_2$ = 0 and $z_1 = z_2$, the application of the Bernoulli equation between points 1 and 2 gives

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad \frac{P_1}{\rho g}+\frac{V_1^2}{2g}=\frac{P_2}{\rho g} \quad\rightarrow\quad V=\sqrt{\frac{2(P_2-P_1)}{\rho_{air}}}$$

where the pressure rise at the tip of the Pitot-static probe is




$$P_2-P_1=\rho_w gh=(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.024\text{ m})\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)$$

$$=235\text{ N/m}^2=\mathbf{235\ Pa}$$

Also, $$\rho_{air}=\frac{P}{RT}=\frac{98\text{ kPa}}{(0.287\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(45+273\text{ K})}=1.074\text{ kg/m}^3$$

Substituting,

$$V_1=\sqrt{\frac{2(235\text{ N/m}^2)}{1.074\text{ kg/m}^3}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)}=\mathbf{20.9\ m/s}$$

***Discussion*** Note that the flow velocity in a pipe or duct can be measured easily by a Pitot-static probe by inserting the probe into the pipe or duct parallel to flow, and reading the differential pressure height. Also note that this is the velocity at the location of the tube. Several readings at several locations in a cross-section may be required to determine the mean flow velocity.

**5-52**

**Solution** The water in an above the ground swimming pool is to be emptied by unplugging the orifice of a horizontal pipe attached to the bottom of the pool. The maximum discharge rate of water is to be determined.

***Assumptions*** **1** The orifice has a smooth entrance, and all frictional losses are negligible. **2** The flow is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable).

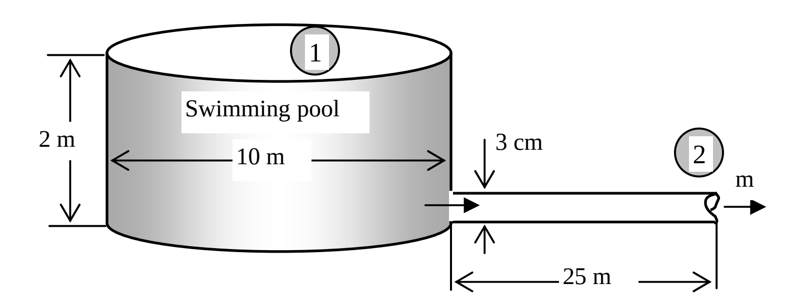
***Analysis*** We take point 1 at the free surface of the pool, and point 2 at the exit of pipe. We take the reference level at the pipe exit ($z_2 = 0$). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the Bernoulli equation between these two points simplifies to

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \quad \rightarrow \quad z_1 = \frac{V_2^2}{2g} \quad \rightarrow \quad V_2 = \sqrt{2gz_1}$$

The maximum discharge rate occurs when the water height in the pool is a maximum, which is the case at the beginning and thus $z_1 = h$. Substituting, the maximum flow velocity and discharge rate become

$$V_{2,\max} = \sqrt{2gh} = \sqrt{2(9.81\,\text{m/s}^2)(2\,\text{m})} = 6.26\,\text{m/s}$$

$$\dot{V}_{\max} = A_{\text{pipe}}V_{2,\max} = \frac{\pi D^2}{4}V_{2,\max} = \frac{\pi(0.03\,\text{m})^2}{4}(6.26\,\text{m/s}) = 0.00443\,\text{m}^3/\text{s} = \mathbf{4.43\,L/s}$$




***Discussion*** The result above is obtained by disregarding all frictional effects. The actual flow rate will be less because of frictional effects during flow and the resulting pressure drop. Also, the flow rate will gradually decrease as the water level in the pipe decreases.

**5-53**

**Solution** The water in an above the ground swimming pool is to be emptied by unplugging the orifice of a horizontal pipe attached to the bottom of the pool. The time it will take to empty the tank is to be determined.

***Assumptions*** **1** The orifice has a smooth entrance, and all frictional losses are negligible. **2** The flow is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable).

***Analysis*** We take point 1 at the free surface of water in the pool, and point 2 at the exit of pipe. We take the reference level at the pipe exit ($z_2 = 0$). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the Bernoulli equation between these two points simplifies to

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad z_1=\frac{V_2^2}{2g} \quad\rightarrow\quad V_2=\sqrt{2gz_1}$$

For generality, we express the water height in the pool at any time $t$ by $z$, and the discharge velocity by $V_2=\sqrt{2gz}$. Note that water surface in the pool moves down as the pool drains, and thus $z$ is a variable whose value changes from $h$ at the beginning to *0* when the pool is emptied completely.

We denote the diameter of the orifice by $D$, and the diameter of the pool by $D_o$. The flow rate of water from the pool is obtained by multiplying the discharge velocity by the orifice cross-sectional area,

$$\dot{V}=A_{\text{orifice}}V_2=\frac{\pi D^2}{4}\sqrt{2gz}$$

Then the amount of water that flows through the orifice during a differential time interval $dt$ is

$$dV=\dot{V}dt=\frac{\pi D^2}{4}\sqrt{2gz}\,dt \qquad (1)$$

which, from conservation of mass, must be equal to the decrease in the volume of water in the pool,

$$dV=A_{\text{tank}}(-dz)=-\frac{\pi D_0^2}{4}dz \qquad (2)$$

where $dz$ is the change in the water level in the pool during $dt$. (Note that $dz$ is a negative quantity since the positive direction of $z$ is upwards. Therefore, we used $-dz$ to get a positive quantity for the amount of water discharged). Setting Eqs. (1) and (2) equal to each other and rearranging,

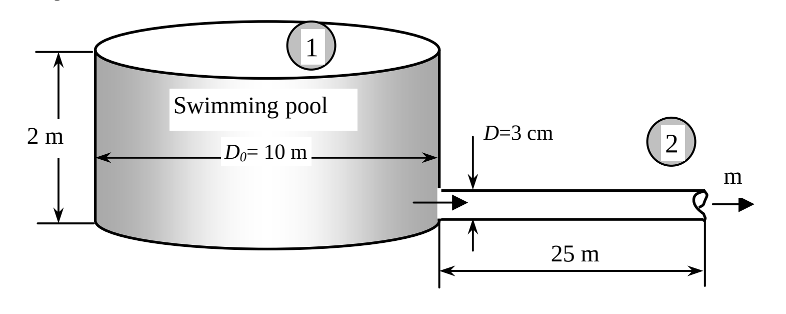
$$\frac{\pi D^2}{4}\sqrt{2gz}\,dt=-\frac{\pi D_0^2}{4}dz \quad\rightarrow\quad dt=-\frac{D_0^2}{D^2}\sqrt{\frac{1}{2gz}}dz=-\frac{D_0^2}{D^2\sqrt{2g}}z^{-\frac{1}{2}}dz$$

The last relation can be integrated easily since the variables are separated. Letting $t_f$ be the discharge time and integrating it from $t = 0$ when $z = h$ to $t = t_f$ when $z = 0$ (completely drained pool) gives

$$\int_{t=0}^{t_f}dt=-\frac{D_0^2}{D^2\sqrt{2g}}\int_{z=z_1}^{0}z^{-1/2}dz \quad\rightarrow\quad t_f=-\frac{D_0^2}{D^2\sqrt{2g}}\left|\frac{z^{\frac{1}{2}}}{\frac{1}{2}}\right|_{z_1}^{0}=\frac{2D_0^2}{D^2\sqrt{2g}}\sqrt{h}=\frac{D_0^2}{D^2}\sqrt{\frac{2h}{g}}$$

Substituting, the draining time of the pool will be

$$t_f=\frac{(10\text{ m})^2}{(0.03\text{ m})^2}\sqrt{\frac{2(2\text{ m})}{9.81\text{ m/s}^2}}=70{,}950\text{ s}=\mathbf{19.7\ h}$$




***Discussion*** This is the minimum discharging time since it is obtained by neglecting all friction; the actual discharging time will be longer. Note that the discharging time is inversely proportional to the square of the orifice diameter. Therefore, the discharging time can be reduced to one-fourth by doubling the diameter of the orifice.



**5-54** 

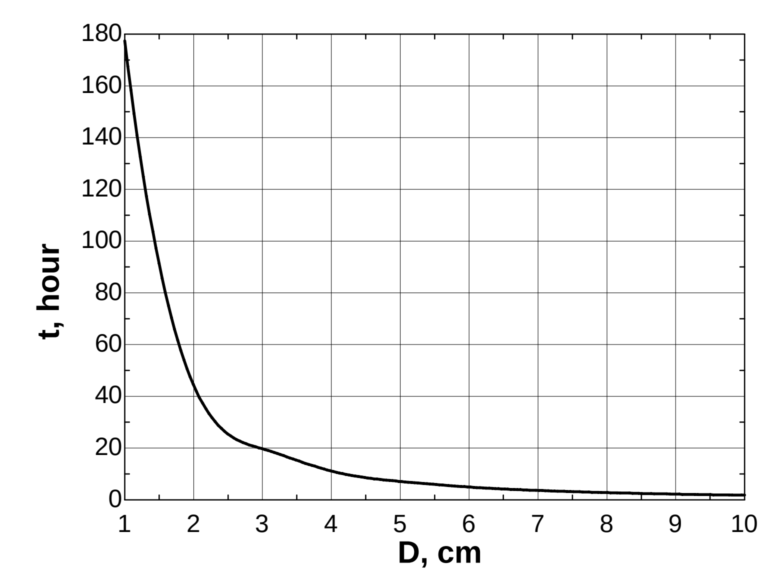
**Solution** The previous problem is reconsidered. The effect of the discharge pipe diameter on the time required to empty the pool completely as the diameter varies from 1 to 10 cm in increments of 1 cm is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
g=9.81 "m/s2"
rho=1000 "kg/m3"
h=2 "m"
D=d_pipe/100 "m"
D_pool=10 "m"

V_initial=SQRT(2*g*h) "m/s"
Ac=pi*D^2/4
V_dot=Ac*V_initial*1000 "m3/s"
t=(D_pool/D)^2*SQRT(2*h/g)/3600
"hour"
```

| Pipe diameter *D*, m | Discharge time *t*, h |
|---|---|
| 1 | 177.4 |
| 2 | 44.3 |
| 3 | 19.7 |
| 4 | 11.1 |
| 5 | 7.1 |
| 6 | 4.9 |
| 7 | 3.6 |
| 8 | 2.8 |
| 9 | 2.2 |
| 10 | 1.8 |




***Discussion*** As you can see from the plot, the discharge time is drastically reduced by increasing the pipe diameter.

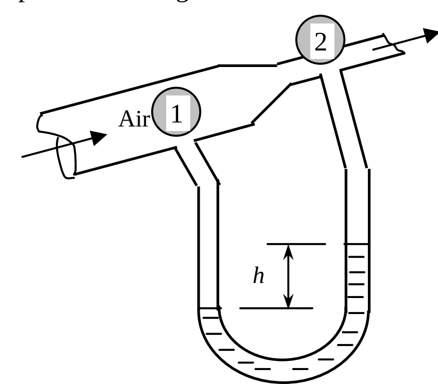
**5-55**

**Solution** Air flows upward at a specified rate through an inclined pipe whose diameter is reduced through a reducer. The differential height between fluid levels of the two arms of a water manometer attached across the reducer is to be determined.

***Assumptions*** **1** The flow through the duct is steady, incompressible and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** Air is an ideal gas. **3** The effect of air column on the pressure change is negligible because of its low density. **4** The air flow is parallel to the entrance of each arm of the manometer, and thus no dynamic effects are involved.

***Properties*** We take the density of water to be $\rho = 1000$ kg/m$^3$. The gas constant of air is $R = 0.287$ kPa·m$^3$/kg·K.

***Analysis*** We take points 1 and 2 at the lower and upper connection points, respectively, of the two arms of the manometer, and take the lower connection point as the reference level. Noting that the effect of elevation on the pressure change of a gas is negligible, the application of the Bernoulli equation between points 1 and 2 gives




$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad P_1-P_2=\rho_{\text{air}}\frac{V_2^2-V_1^2}{2}$$

where $$\rho_{air}=\frac{P}{RT}=\frac{110\text{ kPa}}{(0.287\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(50+273\text{ K})}=1.19\text{ kg/m}^3$$

$$V_1=\frac{\dot V}{A_1}=\frac{\dot V}{\pi D_1^2/4}=\frac{0.045\text{ m}^3/\text{s}}{\pi(0.06\text{ m})^2/4}=15.9\text{ m/s}$$

$$V_2=\frac{\dot V}{A_2}=\frac{\dot V}{\pi D_2^2/4}=\frac{0.045\text{ m}^3/\text{s}}{\pi(0.04\text{ m})^2/4}=35.8\text{ m/s}$$

Substituting,

$$P_1-P_2=(1.19\text{ kg/m}^3)\frac{(35.8\text{ m/s})^2-(15.9\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)=612\text{ N/m}^2=612\text{ Pa}$$

The differential height of water in the manometer corresponding to this pressure change is determined from $\Delta P=\rho_{water}gh$ to be

$$h=\frac{P_1-P_2}{\rho_{water}g}=\frac{612\text{ N/m}^2}{(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)=0.0624\text{ m}=\mathbf{6.24\text{ cm}}$$

***Discussion*** When the effect of air column on pressure change is considered, the pressure change becomes

$$P_1-P_2=\frac{\rho_{air}(V_2^2-V_1^2)}{2}+\rho_{air}g(z_2-z_1)$$

$$=(1.19\text{ kg/m}^3)\left[\frac{(35.8\text{ m/s})^2-(15.9\text{ m/s})^2}{2}+(9.81\text{ m/s}^2)(0.2\text{ m})\right]\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)$$

$$=(612+2)\text{ N/m}^2=614\text{ N/m}^2=614\text{ Pa}$$

This difference between the two results (612 and 614 Pa) is less than 1%. Therefore, the effect of air column on pressure change is, indeed, negligible as assumed. In other words, the pressure change of air in the duct is almost entirely due to velocity change, and the effect of elevation change is negligible. Also, if we were to account for the $\Delta z$ of air flow, then it would be more proper to account for the $\Delta z$ of air in the manometer by using $\rho_{\text{water}}$ - $\rho_{\text{air}}$ instead of $\rho_{\text{water}}$ when calculating $h$. The additional air column in the manometer tends to cancel out the change in pressure due to the elevation difference in the flow in this case.

**5-56E**

**Solution** Air is flowing through a venturi meter with known diameters and measured pressures. A relation for the flow rate is to be obtained, and its numerical value is to be determined.

***Assumptions*** **1**The flow through the venturi is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** The effect of air column on the pressure change is negligible because of its low density, and thus the pressure can be assumed to be uniform at a given cross-section of the venturi meter (independent of elevation change). **3** The flow is horizontal (this assumption is usually unnecessary for gas flow.).

***Properties*** The density of air is given to be $\rho = 0.075$ lbm/ft$^3$.

***Analysis*** We take point 1 at the main flow section and point 2 at the throat along the centerline of the venturi meter. Noting that $z_1 = z_2$, the application of the Bernoulli equation between points 1 and 2 gives

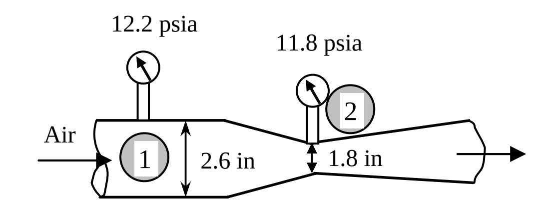
$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad \rightarrow \quad P_1-P_2=\rho\frac{V_2^2-V_1^2}{2} \qquad (1)$$

The flow is assumed to be incompressible and thus the density is constant. Then the conservation of mass relation for this single stream steady flow device can be expressed as

$$\dot V_1=\dot V_2=\dot V \quad \rightarrow \quad A_1V_1=A_2V_2=\dot V \quad \rightarrow \quad V_1=\frac{\dot V}{A_1} \quad \text{and} \quad V_2=\frac{\dot V}{A_2} \qquad (2)$$

Substituting into Eq. (1),

$$P_1-P_2=\rho\frac{(\dot V/A_2)^2-(\dot V/A_1)^2}{2}=\frac{\rho\dot V^2}{2A_2^2}\left(1-\frac{A_2^2}{A_1^2}\right)$$




Solving for $\dot V$ gives the desired relation for the flow rate,

$$\dot V=A_2\sqrt{\frac{2(P_1-P_2)}{\rho[1-(A_2/A_1)^2]}} \qquad (3)$$

The flow rate for the given case can be determined by substituting the given values into this relation to be

$$\dot V=\frac{\pi D_2^2}{4}\sqrt{\frac{2(P_1-P_2)}{\rho[1-(D_2/D_1)^4]}}=\frac{\pi(1.8/12\text{ ft})^2}{4}\sqrt{\frac{2(12.2-11.8)\text{ psi}}{(0.075\text{ lbm/ft}^3)[1-(1.8/2.6)^4]}\left(\frac{144\text{ lbf/ft}^2}{1\text{ psi}}\right)\left(\frac{32.2\text{ lbm}\cdot\text{ft/s}^2}{1\text{ lbf}}\right)}$$

$$= \mathbf{4.48\ ft^3/s}$$

***Discussion*** Venturi meters are commonly used as flow meters to measure the flow rate of gases and liquids by simply measuring the pressure difference $P_1$ - $P_2$ by a manometer or pressure transducers. The actual flow rate will be less than the value obtained from Eq. (3) because of the friction losses along the wall surfaces in actual flow. But this difference can be as little as 1% in a well-designed venturi meter. The effects of deviation from the idealized Bernoulli flow can be accounted for by expressing Eq. (3) as

$$\dot V=C_cA_2\sqrt{\frac{2(P_1-P_2)}{\rho[1-(A_2/A_1)^2]}}$$

where $C_c$ is the *venturi discharge coefficient* whose value is less than 1 (it is as large as 0.99 for well-designed venturi meters in certain ranges of flow). For Re > $10^5$, the value of venturi discharge coefficient is usually greater than 0.96.

**5-57**

**Solution** The gage pressure in the water mains of a city at a particular location is given. It is to be determined if this main can serve water to neighborhoods that are at a given elevation relative to this location.

***Assumptions*** Water is incompressible and thus its density is constant.

***Properties*** We take the density of water to be $\rho$ = 1000 kg/m$^3$.

***Analysis*** Noting that the gage pressure at a dept of *h* in a fluid is given by $P_{gage} = \rho_{water} gh$, the height of a fluid column corresponding to a gage pressure of 400 kPa is determined to be

$$h = \frac{P_{gage}}{\rho_{water} g} = \frac{400{,}000\ \text{N/m}^2}{(1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)}\left(\frac{1\ \text{kg}\cdot\text{m/s}^2}{1\ \text{N}}\right) = 40.8\ \text{m}$$

Water Main, 400 kPa

which is less than 50 m. Therefore, this main **cannot** serve water to neighborhoods that are 50 m above this location.

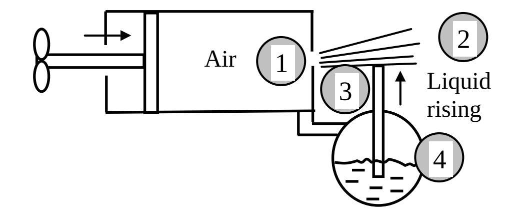
***Discussion*** Note that *h* must be much greater than 50 m for water to have enough pressure to serve the water needs of the neighborhood.

**5-58**

**Solution** A hand-held bicycle pump with a liquid reservoir is used as an atomizer by forcing air at a high velocity through a small hole. The minimum speed that the piston must be moved in the cylinder to initiate the atomizing effect is to be determined.

***Assumptions*** **1**The flows of air and water are steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** Air is an ideal gas. **3** The liquid reservoir is open to the atmosphere. **4** The device is held horizontal. **5** The water velocity through the tube is low.

***Properties*** We take the density of water to be $\rho$ = 1000 kg/m$^3$. The gas constant of air is $R$ = 0.287 kPa·m$^3$/kg·K.




***Analysis*** We take point 1 at the exit of the hole, point 2 in air far from the hole on a horizontal line, point 3 at the exit of the tube in air stream (so that points 1 and 3 coincide), and point 4 at the free surface of the liquid in the reservoir ($P_2 = P_4 = P_{atm}$ and $P_1 = P_3$). We also take the level of the hole to be the reference level (so that $z_1 = z_2 = z_3 = 0$ and $z_4$ = -h). Noting that $V_2 \cong V_3 \cong V_4 \cong 0$, the Bernoulli equation for the air and water streams becomes

$$\textit{Water (3-4)}: \quad \frac{P_3}{\rho g}+\frac{V_3^2}{2g}+z_3=\frac{P_4}{\rho g}+\frac{V_4^2}{2g}+z_4 \;\rightarrow\; \frac{P_1}{\rho g}=\frac{P_{atm}}{\rho g}+(-h) \;\rightarrow\; P_1-P_{atm}=-\rho_{\text{water}}gh \qquad (1)$$

$$\textit{Air (1-2)}: \quad \frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \;\rightarrow\; \frac{P_1}{\rho g}+\frac{V_1^2}{2g}=\frac{P_{atm}}{\rho g} \;\rightarrow\; V_1=\sqrt{\frac{2(P_{atm}-P_1)}{\rho_{\text{air}}}} \qquad (2)$$

where

$$\rho_{air}=\frac{P}{RT}=\frac{95\text{ kPa}}{(0.287\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(20+273\text{ K})}=1.13\text{ kg/m}^3$$

Combining Eqs. (1) and (2) and substituting the numerical values,

$$V_1=\sqrt{\frac{2(P_{atm}-P_1)}{\rho_{\text{air}}}}=\sqrt{\frac{2\rho_{\text{water}}gh}{\rho_{\text{air}}}}=\sqrt{\frac{2(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.1\text{ m})}{1.13\text{ kg/m}^3}}=41.7\text{ m/s}$$

Taking the flow of air to be steady and incompressible, the conservation of mass for air can be expressed as

$$\dot{V}_{\text{piston}}=\dot{V}_{\text{hole}} \;\rightarrow\; V_{\text{piston}}A_{\text{piston}}=V_{\text{hole}}A_{\text{hole}} \;\rightarrow\; V_{\text{piston}}=\frac{A_{\text{hole}}}{A_{\text{piston}}}V_{\text{hole}}=\frac{\pi D_{\text{hole}}^2/4}{\pi D_{\text{piston}}^2/4}V_1$$

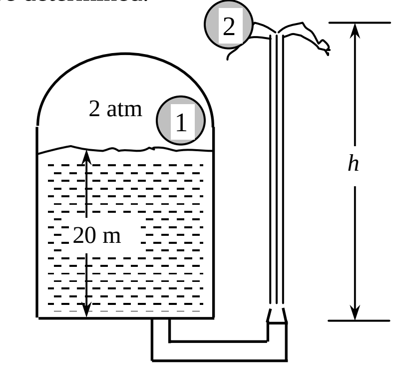
Simplifying and substituting, the piston velocity is determined to be

$$V_{\text{piston}}=\left(\frac{D_{\text{hole}}}{D_{\text{piston}}}\right)^2 V_1=\left(\frac{0.3\text{ cm}}{5\text{ cm}}\right)^2(41.7\text{ m/s})=\mathbf{0.15\text{ m/s}}$$

***Discussion*** In reality, the piston velocity must be higher to overcome the losses. Also, a lower piston velocity will do the job if the diameter of the hole is reduced.

**5-59**

**Solution** The water height in an airtight pressurized tank is given. A hose pointing straight up is connected to the bottom of the tank. The maximum height to which the water stream could rise is to be determined.




***Assumptions*** **1** The flow is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** The friction between the water and air is negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** We take point 1 at the free surface of water in the tank, and point 2 at the top of the water trajectory. Also, we take the reference level at the bottom of the tank. At the top of the water trajectory $\mathbf{V}_2 = 0$, and atmospheric pressure pertains. Noting that $z_1 = 20$ m, $P_{1,gage} = 2$ atm, $P_2 = P_{atm}$, and that the fluid velocity at the free surface of the tank is very low ($V_1 \cong 0$), the Bernoulli equation between these two points simplifies to

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad \frac{P_1}{\rho g}+z_1=\frac{P_{atm}}{\rho g}+z_2 \quad\rightarrow\quad z_2=\frac{P_1-P_{atm}}{\rho g}+z_1=\frac{P_{1,\text{gage}}}{\rho g}+z_1$$

Substituting,

$$z_2=\frac{2\text{ atm}}{(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)}\left(\frac{101{,}325\text{ N/m}^2}{1\text{ atm}}\right)\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)+20=\mathbf{40.7\text{ m}}$$

Therefore, the water jet can rise as high as 40.7 m into the sky from the ground.

***Discussion*** The result obtained by the Bernoulli equation represents the upper limit, and should be interpreted accordingly. It tells us that the water cannot possibly rise more than 40.7 m (giving us an upper limit), and in all likelihood, the rise will be much less because of frictional losses.

**5-60**

**Solution** A Pitot-static probe equipped with a water manometer is held parallel to air flow, and the differential height of the water column is measured. The flow velocity of air is to be determined.

***Assumptions*** **1**The flow of air is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** The effect of air column on the pressure change is negligible because of its low density, and thus the air column in the manometer can be ignored.

***Properties*** We take the density of water to be $\rho$ = 1000 kg/m$^3$. The density of air is given to be 1.25 kg/m$^3$.

***Analysis*** We take point 1 on the side of the probe where the entrance is parallel to flow and is connected to the static arm of the Pitot-static probe, and point 2 at the tip of the probe where the entrance is normal to flow and is connected to the dynamic arm of the Pitot-static probe. Noting that point 2 is a stagnation point and thus $V_2$ = 0 and $z_1 = z_2$, the application of the Bernoulli equation between points 1 and 2 gives

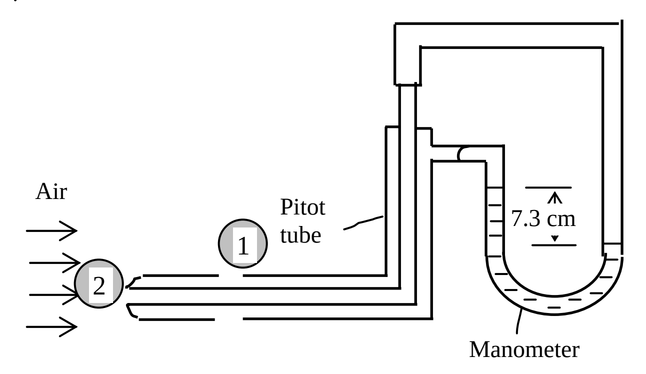
$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad \frac{P_1}{\rho g}+\frac{V_1^2}{2g}=\frac{P_2}{\rho g} \quad\rightarrow\quad V_1=\sqrt{\frac{2(P_2-P_1)}{\rho_{air}}} \qquad (1)$$

The pressure rise at the tip of the Pitot-static probe is simply the pressure change indicated by the differential water column of the manometer,

$$P_2-P_1=\rho_{\text{water}}gh \qquad (2)$$

Combining Eqs. (1) and (2) and substituting, the flow velocity is determined to be

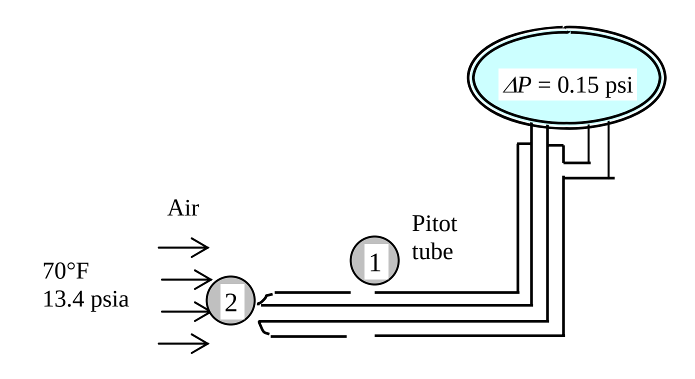
$$V_1=\sqrt{\frac{2\rho_{\text{water}}gh}{\rho_{\text{air}}}}=\sqrt{\frac{2(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.073\text{ m})}{1.25\text{ kg/m}^3}}=\mathbf{33.8\text{ m/s}}$$




***Discussion*** Note that flow velocity in a pipe or duct can be measured easily by a Pitot-static probe by inserting the probe into the pipe or duct parallel to flow, and reading the differential height. Also note that this is the velocity at the location of the tube. Several readings at several locations in a cross-section may be required to determine the mean flow velocity.

**5-61E**

**Solution** A Pitot-static probe equipped with a differential pressure gage is used to measure the air velocity in a duct. For a given differential pressure reading, the flow velocity of air is to be determined.




***Assumptions*** The flow of air is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable).

***Properties*** The gas constant of air is $R$ = 0.3704 psia·ft$^3$/lbm·R.

***Analysis*** We take point 1 on the side of the probe where the entrance is parallel to flow and is connected to the static arm of the Pitot-static probe, and point 2 at the tip of the probe where the entrance is normal to flow and is connected to the dynamic arm of the Pitot-static probe. Noting that point 2 is a stagnation point and thus $V_2 = 0$ and $z_1 = z_2$, the application of the Bernoulli equation between points 1 and 2 gives

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad \frac{P_1}{\rho g}+\frac{V_1^2}{2g}=\frac{P_2}{\rho g} \quad\rightarrow\quad V_1=\sqrt{\frac{2(P_2-P_1)}{\rho}}$$

where

$$\rho=\frac{P}{RT}=\frac{13.4\text{ psia}}{(0.3704\text{ psia}\cdot\text{ft}^3/\text{lbm}\cdot\text{R})(70+460\text{ R})}=0.0683\text{ lbm/ft}^3$$

Substituting the given values, the flow velocity is determined to be

$$V_1=\sqrt{\frac{2(0.15\text{ psi})}{0.0683\text{ lbm/ft}^3}\left(\frac{144\text{ lbf/ft}^2}{1\text{ psi}}\right)\left(\frac{32.2\text{ lbm}\cdot\text{ft/s}^2}{1\text{ lbf}}\right)}=\mathbf{143\ ft/s}$$

***Discussion*** Note that flow velocity in a pipe or duct can be measured easily by a Pitot-static probe by inserting the probe into the pipe or duct parallel to flow, and reading the pressure differential. Also note that this is the velocity at the location of the tube. Several readings at several locations in a cross-section may be required to determine the mean flow velocity.

**5-62**

**Solution** In a power plant, water enters the nozzles of a hydraulic turbine at a specified pressure. The maximum velocity water can be accelerated to by the nozzles is to be determined.

***Assumptions*** **1**The flow of water is steady, incompressible, and irrotational with negligible frictional effects (so that the Bernoulli equation is applicable). **2** Water enters the nozzle with a low velocity.

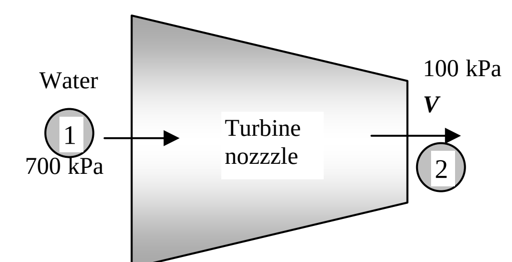
***Properties*** We take the density of water to be $\rho = 1000$ kg/m$^3$.

***Analysis*** We take points 1 and 2 at the inlet and exit of the nozzle, respectively. Noting that $V_1 \cong 0$ and $z_1 = z_2$, the application of the Bernoulli equation between points 1 and 2 gives

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad \to \quad \frac{P_1}{\rho g}=\frac{P_{atm}}{\rho g}+\frac{V_2^2}{2g} \quad \to \quad V_2=\sqrt{\frac{2(P_1-P_{atm})}{\rho}}$$

Substituting the given values, the nozzle exit velocity is determined to be

$$V_1=\sqrt{\frac{2(700-100)\text{ kPa}}{1000\text{ kg/m}^3}\left(\frac{1000\text{ N/m}^2}{1\text{ kPa}}\right)\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)}=\mathbf{34.6\text{ m/s}}$$




***Discussion*** This is the maximum nozzle exit velocity, and the actual velocity will be less because of friction between water and the walls of the nozzle.

---

## Energy Equation

---

**5-63C**

**Solution** We are to analyze whether temperature can decrease during steady adiabatic flow of an incompressible fluid.

***Analysis*** **It is *impossible* for the fluid temperature to decrease** during steady, incompressible, adiabatic flow of an incompressible fluid, since this would require the entropy of an adiabatic system to decrease, which would be a violation of the 2$^{nd}$ law of thermodynamics.

***Discussion*** The entropy of a fluid *can* decrease, but only if we remove heat.

---

**5-64C**

**Solution** We are to determine if frictional effects are negligible in the steady adiabatic flow of an incompressible fluid if the temperature remains constant.

***Analysis*** **Yes, the *frictional effects* are negligible** if the fluid temperature remains constant during steady, incompressible flow since any irreversibility such as friction would cause the entropy and thus temperature of the fluid to increase during adiabatic flow.

***Discussion*** Thus, this scenario would never occur in real life since all fluid flows have frictional effects.

---

**5-65C**

**Solution** We are to define and discuss irreversible head loss.

***Analysis*** *Irreversible head loss* is the **loss of mechanical energy due to irreversible processes** (such as friction) **in piping expressed as an equivalent column height of fluid**, i.e., head. Irreversible head loss is related to the mechanical energy loss in piping by $\boxed{h_L = \dfrac{e_{\text{mech loss, piping}}}{g} = \dfrac{\dot{E}_{\text{mech loss, piping}}}{\dot{m}g}}$.

***Discussion*** $h_L$ is always positive. It can never be negative, since this would violate the second law of thermodynamics.

**5-66C**

**Solution** We are to define and discuss useful pump head.

***Analysis*** *Useful pump head* is the **useful power input to the pump expressed as an equivalent column height of fluid**. It is related to the useful pumping power input by $\boxed{h_{\text{pump}} = \dfrac{w_{\text{pump, u}}}{g} = \dfrac{\dot{W}_{\text{pump, u}}}{\dot{m}g}}$.

***Discussion*** Part of the power supplied to the pump is *not* useful, but rather is wasted because of irreversible losses in the pump. This is the reason that pumps have a pump efficiency that is always less than one.

**5-67C**

**Solution** We are to define and discuss the kinetic energy correction factor.

***Analysis*** The *kinetic energy correction factor* is a **correction factor to account for the fact that kinetic energy using average velocity is not the same as the actual kinetic energy using the actual velocity profile** (the square of a sum is not equal to the sum of the squares of its components). The effect of kinetic energy factor is usually negligible, especially for turbulent pipe flows. However, for laminar pipe flows, the effect of $\alpha$ is sometimes significant.

***Discussion*** Even though the effect of ignoring $\alpha$ is usually insignificant, it is wise to keep $\alpha$ in our analyses to increase accuracy and so that we do not forget about it in situations where it *is* significant, such as in some laminar pipe flows.

**5-68C**

**Solution** We are to analyze the cause of some strange behavior of a water jet.

***Analysis*** The problem does not state whether the water in the tank is open to the atmosphere or not. Let's assume that the water surface *is* exposed to atmospheric pressure. By the Bernoulli equation, the maximum theoretical height to which the water stream could rise is the tank water level, which is 20 meters above the ground. Since the water rises *above* the tank level, the tank cover must be airtight, containing pressurized air above the water surface. In other words, the water in the tank is *not* exposed to atmospheric pressure.

***Discussion*** Alternatively, a pump would have to pressurize the water somewhere in the hose.

**5-69**

**Solution** Underground water is pumped to a pool at a given elevation. The maximum flow rate and the pressures at the inlet and outlet of the pump are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The elevation difference between the inlet and the outlet of the pump is negligible. **3** We assume the frictional effects in piping to be negligible since the *maximum* flow rate is to be determined, $\dot{E}_{\text{mech loss, pipping}} = 0$. **4** The effect of the kinetic energy correction factors is negligible, $\alpha = 1$.

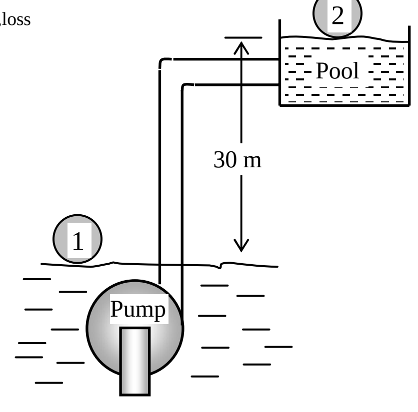
***Properties*** We take the density of water to be 1 kg/L = 1000 kg/m$^3$.

***Analysis*** (*a*) The pump-motor draws 3-kW of power, and is 70% efficient. Then the useful mechanical (shaft) power it delivers to the fluid is

$$\dot{W}_{\text{pump, u}} = \eta_{\text{pump-motor}}\dot{W}_{\text{electric}} = (0.70)(3\,\text{kW}) = 2.1\,\text{kW}$$

We take point 1 at the free surface of underground water, which is also taken as the reference level ($z_1 = 0$), and point 2 at the free surface of the pool. Also, both 1 and 2 are open to the atmosphere ($P_1 = P_2 = P_{\text{atm}}$), the velocities are negligible at both points ($V_1 \cong V_2 \cong 0$), and frictional losses in piping are disregarded. Then the energy equation for steady incompressible flow through a control volume between these two points that includes the pump and the pipes reduces to

$$\dot{m}\left(\frac{P_1}{\rho} + \alpha_1\frac{V_1^2}{2} + gz_1\right) + \dot{W}_{\text{pump}} = \dot{m}\left(\frac{P_2}{\rho} + \alpha_2\frac{V_2^2}{2} + gz_2\right) + \dot{W}_{\text{turbine}} + \dot{E}_{\text{mech,loss}}$$




In the absence of a turbine, $\dot{E}_{\text{mech, loss}} = \dot{E}_{\text{mech loss, pump}} + \dot{E}_{\text{mech loss, piping}}$ and

$\dot{W}_{\text{pump, u}} = \dot{W}_{\text{pump}} - \dot{E}_{\text{mech loss, pump}}$.

Thus, $\dot{W}_{\text{pump, u}} = \dot{m}gz_2$.

Then the mass and volume flow rates of water become

$$\dot{m} = \frac{\dot{W}_{\text{pump, u}}}{gz_2} = \frac{2.1\,\text{kJ/s}}{(9.81\,\text{m/s}^2)(30\,\text{m})}\left(\frac{1000\,\text{m}^2/\text{s}^2}{1\,\text{kJ}}\right) = 7.14\,\text{kg/s}$$

$$\dot{V} = \frac{\dot{m}}{\rho} = \frac{7.14\,\text{kg/s}}{1000\,\text{kg/m}^3} = \mathbf{7.14\times10^{-3}\,m^3/s}$$

(*b*) We take points 3 and 4 at the inlet and the exit of the pump, respectively, where the flow velocities are

$$V_3 = \frac{\dot{V}}{A_3} = \frac{\dot{V}}{\pi D_3^2/4} = \frac{7.14\times10^{-3}\,\text{m}^3/\text{s}}{\pi(0.07\,\text{m})^2/4} = 1.86\,\text{m/s}, \quad V_4 = \frac{\dot{V}}{A_4} = \frac{\dot{V}}{\pi D_4^2/4} = \frac{7.14\times10^{-3}\,\text{m}^3/\text{s}}{\pi(0.05\,\text{m})^2/4} = 3.64\,\text{m/s}$$

We take the pump as the control volume. Noting that $z_3 = z_4$, the energy equation for this control volume reduces to

$$\dot{m}\left(\frac{P_3}{\rho} + \alpha_3\frac{V_3^2}{2} + gz_3\right) + \dot{W}_{\text{pump}} = \dot{m}\left(\frac{P_4}{\rho} + \alpha_4\frac{V_4^2}{2} + gz_4\right) + \dot{W}_{\text{turbine}} + \dot{E}_{\text{mech loss, pump}} \rightarrow P_4 - P_3 = \frac{\rho\alpha(V_3^2 - V_4^2)}{2} + \frac{\dot{W}_{\text{pump,u}}}{\dot{V}}$$

Substituting,

$$P_4 - P_3 = \frac{(1000\,\text{kg/m}^3)(1.0)\left[(1.86\,\text{m/s})^2 - (3.64\,\text{m/s})^2\right]}{2}\left(\frac{1\,\text{kN}}{1000\,\text{kg}\cdot\text{m/s}^2}\right) + \frac{2.1\,\text{kJ/s}}{7.14\times10^{-3}\,\text{m}^3/\text{s}}\left(\frac{1\,\text{kN}\cdot\text{m}}{1\,\text{kJ}}\right)$$

$$= (-4.9 + 294.1)\,\text{kN/m}^2 = 289.2\,\text{kPa} \cong \mathbf{289\,kPa}$$

***Discussion*** In an actual system, the flow rate of water will be less because of friction in the pipes. Also, the effect of flow velocities on the pressure change across the pump is negligible in this case (under 2%) and can be ignored.

**5-70**

**Solution** Underground water is pumped to a pool at a given elevation. For a given head loss, the flow rate and the pressures at the inlet and outlet of the pump are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The elevation difference between the inlet and the outlet of the pump is negligible. **3** The effect of the kinetic energy correction factors is negligible, $\alpha = 1$.

***Properties*** We take the density of water to be 1 kg/L = 1000 kg/m$^3$.

***Analysis*** (*a*) The pump-motor draws 3-kW of power, and is 70% efficient. Then the useful mechanical (shaft) power it delivers to the fluid is

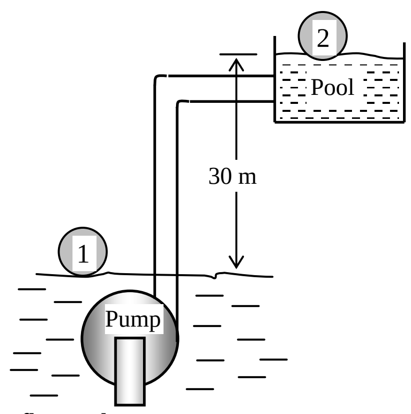
$$\dot{W}_{\text{pump, u}} = \eta_{\text{pump-motor}} \dot{W}_{\text{electric}} = (0.70)(3\,\text{kW}) = 2.1\,\text{kW}$$

We take point 1 at the free surface of underground water, which is also taken as the reference level ($z_1 = 0$), and point 2 at the free surface of the pool. Also, both 1 and 2 are open to the atmosphere ($P_1 = P_2 = P_{atm}$), and the velocities are negligible at both points ($V_1 \cong V_2 \cong 0$). Then the energy equation for steady incompressible flow through a control volume between these two points that includes the pump and the pipes reduces to

$$\dot{m}\left(\frac{P_1}{\rho} + \alpha_1 \frac{V_1^2}{2} + gz_1\right) + \dot{W}_{\text{pump}} = \dot{m}\left(\frac{P_2}{\rho} + \alpha_2 \frac{V_2^2}{2} + gz_2\right) + \dot{W}_{\text{turbine}} + \dot{E}_{\text{mech,loss}}$$

In the absence of a turbine, $\dot{E}_{\text{mech, loss}} = \dot{E}_{\text{mech loss, pump}} + \dot{E}_{\text{mech loss, piping}}$ and $\dot{W}_{\text{pump, u}} = \dot{W}_{\text{pump}} - \dot{E}_{\text{mech loss, pump}}$ and thus

$$\dot{W}_{\text{pump, u}} = \dot{m}gz_2 + \dot{E}_{\text{mech loss, piping}}$$




Noting that $\dot{E}_{\text{mech, loss}} = \dot{m}gh_L$, the mass and volume flow rates of water become

$$\dot{m} = \frac{\dot{W}_{\text{pump, u}}}{gz_2 + gh_L} = \frac{\dot{W}_{\text{pump, u}}}{g(z_2 + h_L)}$$

$$= \frac{2.1\,\text{kJ/s}}{(9.81\,\text{m/s}^2)(30+5\,\text{m})}\left(\frac{1000\,\text{m}^2/\text{s}^2}{1\,\text{kJ}}\right) = 6.116\,\text{kg/s}$$

$$\dot{V} = \frac{\dot{m}}{\rho} = \frac{6.116\,\text{kg/s}}{1000\,\text{kg/m}^3} = 6.116 m^3/s \cong \mathbf{6.12 \times 10^{-3}\ m^3/s}$$

(*b*) We take points 3 and 4 at the inlet and the exit of the pump, respectively, where the flow velocities are

$$V_3 = \frac{\dot{V}}{A_3} = \frac{\dot{V}}{\pi D_3^2/4} = \frac{6.116\times10^{-3}\,\text{m}^3/\text{s}}{\pi(0.07\,\text{m})^2/4} = 1.589\,\text{m/s}\,, \quad V_4 = \frac{\dot{V}}{A_4} = \frac{\dot{V}}{\pi D_4^2/4} = \frac{6.116\times10^{-3}\,\text{m}^3/\text{s}}{\pi(0.05\,\text{m})^2/4} = 3.115\,\text{m/s}$$

We take the pump as the control volume. Noting that $z_3 = z_4$, the energy equation for this control volume reduces to

$$\dot{m}\left(\frac{P_3}{\rho} + \alpha_3 \frac{V_3^2}{2} + gz_3\right) + \dot{W}_{\text{pump}} = \dot{m}\left(\frac{P_4}{\rho} + \alpha_4 \frac{V_4^2}{2} + gz_4\right) + \dot{W}_{\text{turbine}} + \dot{E}_{\text{mech loss, pump}} \rightarrow P_4 - P_3 = \frac{\rho\alpha(V_3^2 - V_4^2)}{2} + \frac{\dot{W}_{\text{pump,u}}}{\dot{V}}$$

Substituting,

$$P_4 - P_3 = \frac{(1000\,\text{kg/m}^3)(1.0)\left[(1.589\,\text{m/s})^2 - (3.115\,\text{m/s})^2\right]}{2}\left(\frac{1\,\text{kN}}{1000\,\text{kg}\cdot\text{m/s}^2}\right) + \frac{2.1\,\text{kJ/s}}{6.116\times10^{-3}\,\text{m}^3/\text{s}}\left(\frac{1\,\text{kN}\cdot\text{m}}{1\,\text{kJ}}\right)$$

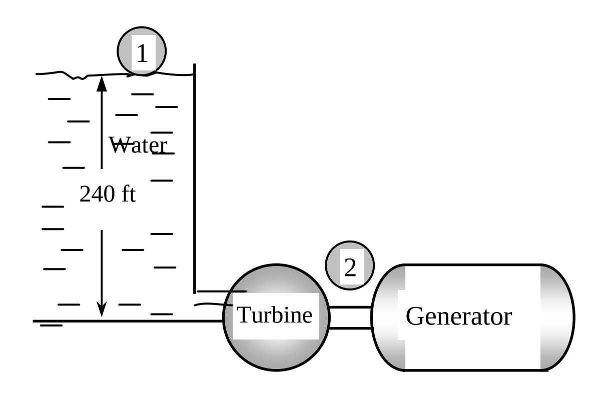
$$= (-3.6 + 343.4)\,\text{kN/m}^2 = 339.8\,\text{kPa} \cong \mathbf{340\ kPa}$$

***Discussion*** Note that frictional losses in the pipes causes the flow rate of water to decrease. Also, the effect of flow velocities on the pressure change across the pump is negligible in this case (about 1%) and can be ignored.

**5-71E**

**Solution** In a hydroelectric power plant, the elevation difference, the power generation, and the overall turbine-generator efficiency are given. The minimum flow rate required is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The water levels at the reservoir and the discharge site remain constant. **3** We assume the flow to be *frictionless* since the *minimum* flow rate is to be determined, $\dot{E}_{\text{mech,loss}} = 0$.




***Properties*** We take the density of water to be $\rho$ = 62.4 lbm/ft$^3$.

***Analysis*** We take point 1 at the free surface of the reservoir and point 2 at the free surface of the discharge water stream, which is also taken as the reference level ($z_2$ = 0). Also, both 1 and 2 are open to the atmosphere ($P_1 = P_2 = P_{\text{atm}}$), the velocities are negligible at both points ($V_1 = V_2 = 0$), and frictional losses are disregarded. Then the energy equation in terms of heads for steady incompressible flow through a control volume between these two points that includes the turbine and the pipes reduces to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad h_{\text{turbine, e}} = z_1$$

Substituting and noting that $\dot{W}_{\text{turbine, elect}} = \eta_{\text{turbine-gen}} \dot{m} g h_{\text{turbine, e}}$, the extracted turbine head and the mass and volume flow rates of water are determined to be

$$h_{\text{turbine, e}} = z_1 = 240 \text{ ft}$$

$$\dot{m} = \frac{\dot{W}_{\text{turbine,elect}}}{\eta_{\text{turbine-gen}} g h_{\text{turbine}}} = \frac{100 \text{ kW}}{0.83(32.2 \text{ ft/s}^2)(240 \text{ ft})}\left(\frac{25{,}037 \text{ ft}^2/\text{s}^2}{1 \text{ Btu/lbm}}\right)\left(\frac{0.9478 \text{ Btu/s}}{1 \text{ kW}}\right) = \mathbf{370\ lbm/s}$$

$$\dot{V} = \frac{\dot{m}}{\rho} = \frac{370 \text{ lbm/s}}{62.4 \text{ lbm/ft}^3} = \mathbf{5.93\ ft^3/s}$$

Therefore, the flow rate of water must be at least 5.93 ft$^3$/s to generate the desired electric power while overcoming friction losses in pipes.

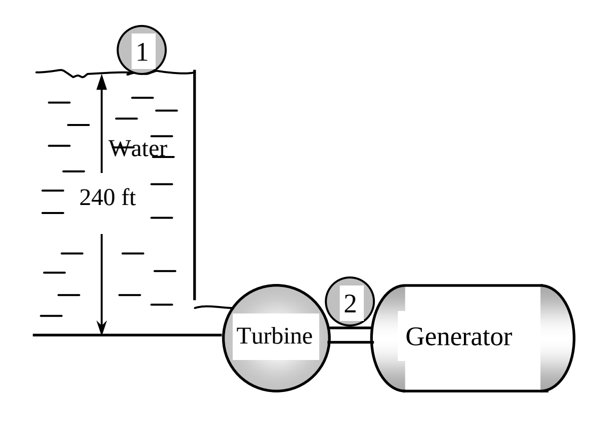
***Discussion*** In an actual system, the flow rate of water will be more because of frictional losses in pipes.

**5-72E**

**Solution** In a hydroelectric power plant, the elevation difference, the head loss, the power generation, and the overall turbine-generator efficiency are given. The flow rate required is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The water levels at the reservoir and the discharge site remain constant.

***Properties*** We take the density of water to be $\rho$ = 62.4 lbm/ft$^3$.




***Analysis*** We take point 1 at the free surface of the reservoir and point 2 at the free surface of the discharge water stream, which is also taken as the reference level ($z_2$ = 0). Also, both 1 and 2 are open to the atmosphere ($P_1 = P_2 = P_{atm}$), the velocities are negligible at both points ($V_1 = V_2 = 0$). Then the energy equation in terms of heads for steady incompressible flow through a control volume between these two points that includes the turbine and the pipes reduces to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad h_{\text{turbine, e}}=z_1-h_L$$

Substituting and noting that $\dot{W}_{\text{turbine, elect}}=\eta_{\text{turbine-gen}}\dot{m}gh_{\text{turbine, e}}$, the extracted turbine head and the mass and volume flow rates of water are determined to be

$$h_{\text{turbine, e}}=z_1-h_L=240-36=204\text{ ft}$$

$$\dot{m}=\frac{\dot{W}_{\text{turbine,elect}}}{\eta_{\text{turbine-gen}}gh_{\text{turbine}}}=\frac{100\text{ kW}}{0.83(32.2\text{ ft/s}^2)(204\text{ ft})}\left(\frac{25{,}037\text{ ft}^2/\text{s}^2}{1\text{ Btu/lbm}}\right)\left(\frac{0.9478\text{ Btu/s}}{1\text{ kW}}\right)=\mathbf{435\text{ lbm/s}}$$

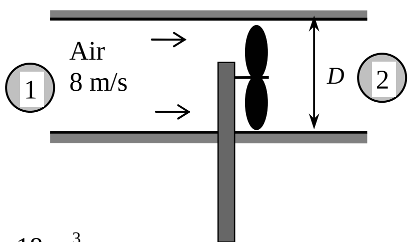
$$\dot{V}=\frac{\dot{m}}{\rho}=\frac{435\text{ lbm/s}}{62.4\text{ lbm/ft}^3}=\mathbf{6.98\text{ ft}^3/\text{s}}$$

Therefore, the flow rate of water must be at least 6.98 ft$^3$/s to generate the desired electric power while overcoming friction losses in pipes.

***Discussion*** Note that the effect of frictional losses in the pipes is to increase the required flow rate of water to generate a specified amount of electric power.

**5-73** **[***Also solved using EES on enclosed DVD***]**

**Solution** A fan is to ventilate a bathroom by replacing the entire volume of air once every 10 minutes while air velocity remains below a specified value. The wattage of the fan-motor unit, the diameter of the fan casing, and the pressure difference across the fan are to be determined.




***Assumptions*** **1** The flow is steady and incompressible. **2** Frictional losses along the flow (other than those due to the fan-motor inefficiency) are negligible. **3** The fan unit is horizontal so that $z$ = constant along the flow (or, the elevation effects are negligible because of the low density of air). **4** The effect of the kinetic energy correction factors is negligible, $\alpha = 1$.

***Properties*** The density of air is given to be 1.25 kg/m$^3$.

***Analysis*** (*a*) The volume of air in the bathroom is $V$ = 2 m × 3 m × 3 m = 18 m$^3$. Then the volume and mass flow rates of air through the casing must be

$$\dot{V} = \frac{V}{\Delta t} = \frac{18\ \text{m}^3}{10 \times 60\ \text{s}} = 0.03\ \text{m}^3/\text{s}$$

$$\dot{m} = \rho \dot{V} = (1.25\ \text{kg/m}^3)(0.03\ \text{m}^3/\text{s}) = 0.0375\ \text{kg/s}$$

We take points 1 and 2 on the inlet and exit sides of the fan, respectively. Point 1 is sufficiently far from the fan so that $P_1 = P_{atm}$ and the flow velocity is negligible ($V_1 = 0$). Also, $P_2 = P_{atm}$. Then the energy equation for this control volume between the points 1 and 2 reduces to

$$\dot{m}\left(\frac{P_1}{\rho} + \alpha_1 \frac{V_1^2}{2} + gz_1\right) + \dot{W}_{\text{pump}} = \dot{m}\left(\frac{P_2}{\rho} + \alpha_2 \frac{V_2^2}{2} + gz_2\right) + \dot{W}_{\text{turbine}} + \dot{E}_{\text{mech,loss}} \quad \rightarrow \quad \dot{W}_{\text{fan, u}} = \dot{m}\alpha_2 \frac{V_2^2}{2}$$

since $\dot{E}_{\text{mech, loss}} = \dot{E}_{\text{mech loss, pump}}$ in this case and $\dot{W}_{\text{pump, u}} = \dot{W}_{\text{pump}} - \dot{E}_{\text{mech loss, pump}}$. Substituting,

$$\dot{W}_{\text{fan, u}} = \dot{m}\alpha_2 \frac{V_2^2}{2} = (0.0375\ \text{kg/s})(1.0)\frac{(8\ \text{m/s})^2}{2}\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{W}}{1\ \text{N}\cdot\text{m/s}}\right) = 1.2\ \text{W}$$

and $$\dot{W}_{\text{fan, elect}} = \frac{\dot{W}_{\text{fan, u}}}{\eta_{\text{fan-motor}}} = \frac{1.2\ \text{W}}{0.5} = \mathbf{2.4\ W}$$

Therefore, the electric power rating of the fan/motor unit must be 2.4 W.

(*b*) For air mean velocity to remain below the specified value, the diameter of the fan casing should be

$$\dot{V} = A_2 V_2 = (\pi D_2^2/4)V_2 \quad \rightarrow \quad D_2 = \sqrt{\frac{4\dot{V}}{\pi V_2}} = \sqrt{\frac{4(0.03\ \text{m}^3/\text{s})}{\pi(8\ \text{m/s})}} = 0.069\ \text{m} = \mathbf{6.9\ cm}$$

(*c*) To determine the pressure difference across the fan unit, we take points 3 and 4 to be on the two sides of the fan on a horizontal line. Noting that $z_3 = z_4$ and $V_3 = V_4$ since the fan is a narrow cross-section and neglecting flow loses (other than the loses of the fan unit, which is accounted for by the efficiency), the energy equation for the fan section reduces to

$$\dot{m}\frac{P_3}{\rho} + \dot{W}_{\text{fan, u}} = \dot{m}\frac{P_4}{\rho} \quad \rightarrow \quad P_4 - P_3 = \frac{\dot{W}_{\text{fan, u}}}{\dot{m}/\rho} = \frac{\dot{W}_{\text{fan, u}}}{\dot{V}}$$

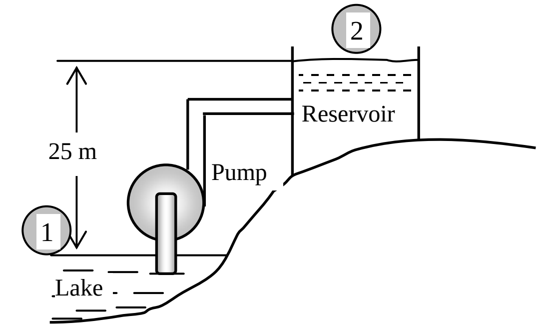
Substituting, $$P_4 - P_3 = \frac{1.2\ \text{W}}{0.03\ \text{m}^3/\text{s}}\left(\frac{1\ \text{N}\cdot\text{m/s}}{1\ \text{W}}\right) = 40\ \text{N/m}^2 = \mathbf{40\ Pa}$$

Therefore, the fan will raise the pressure of air by 40 Pa before discharging it.

***Discussion*** Note that only half of the electric energy consumed by the fan-motor unit is converted to the mechanical energy of air while the remaining half is converted to heat because of imperfections.

**5-74**

**Solution** Water is pumped from a large lake to a higher reservoir. The head loss of the piping system is given. The mechanical efficiency of the pump is to be determined.




***Assumptions*** **1** The flow is steady and incompressible. **2** The elevation difference between the lake and the reservoir is constant.

***Properties*** We take the density of water to be $\rho$ = 1000 kg/m³.

***Analysis*** We choose points 1 and 2 at the free surfaces of the lake and the reservoir, respectively, and take the surface of the lake as the reference level ($z_1$ = 0). Both points are open to the atmosphere ($P_1 = P_2 = P_{atm}$) and the velocities at both locations are negligible ($V_1 = V_2$ = 0). Then the energy equation for steady incompressible flow through a control volume between these two points that includes the pump and the pipes reduces to

$$\dot{m}\left(\frac{P_1}{\rho}+\alpha_1\frac{V_1^2}{2}+gz_1\right)+\dot{W}_{\text{pump}}=\dot{m}\left(\frac{P_2}{\rho}+\alpha_2\frac{V_2^2}{2}+gz_2\right)+\dot{W}_{\text{turbine}}+\dot{E}_{\text{mech,loss}} \rightarrow \dot{W}_{\text{pump,u}}=\dot{m}gz_2+\dot{E}_{\text{mech loss, piping}}$$

since, in the absence of a turbine, $\dot{E}_{\text{mech, loss}}=\dot{E}_{\text{mech loss, pump}}+\dot{E}_{\text{mech loss, piping}}$ and $\dot{W}_{\text{pump, u}}=\dot{W}_{\text{pump}}-\dot{E}_{\text{mech loss, pump}}$. Noting that $\dot{E}_{\text{mech loss, piping}}=\dot{m}gh_L$, the useful pump power is

$$\dot{W}_{\text{pump,u}}=\dot{m}gz_2+\dot{m}gh_L=\rho\dot{V}g(z_2+h_L)$$

$$=(1000\text{ kg/m}^3)(0.025\text{ m}^3\text{/s})(9.81 m/s^2)[(25+7)\text{ m}]\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)$$

$$=7.85\text{ kNm/s}=7.85\text{ kW}$$

Then the mechanical efficiency of the pump becomes

$$\eta_{\text{pump}}=\frac{\dot{W}_{\text{pump, u}}}{\dot{W}_{\text{shaft}}}=\frac{7.85\text{ kW}}{10\text{ kW}}=0.785=\mathbf{78.5\%}$$

***Discussion*** A more practical measure of performance of the pump is the overall efficiency, which can be obtained by multiplying the pump efficiency by the motor efficiency.



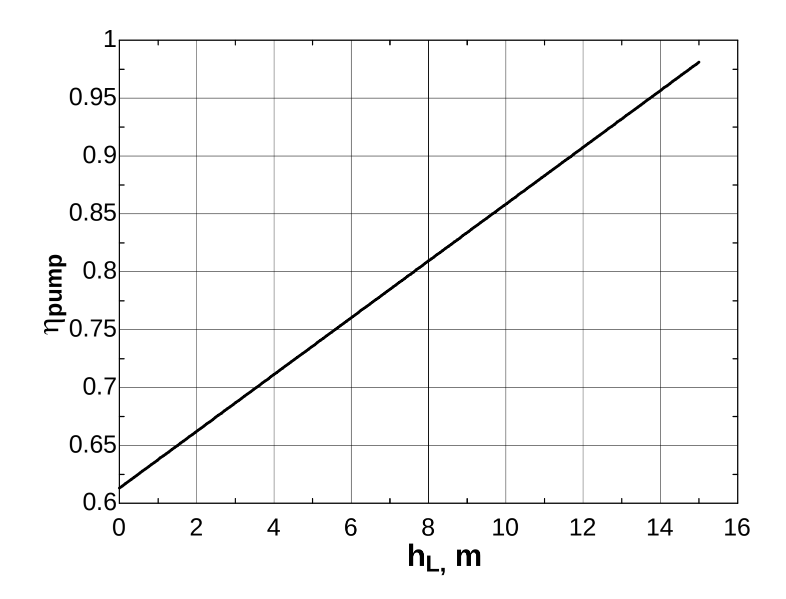

**5-75**

**Solution** The previous problem is reconsidered. The effect of head loss on mechanical efficiency of the pump. as the head loss varies 0 to 20 m in increments of 2 m is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
g=9.81 "m/s2"
rho=1000 "kg/m3"
z2=25 "m"
W_shaft=10 "kW"
V_dot=0.025 "m3/s"
W_pump_u=rho*V_dot*g*(z2+h_L)/1000 "kW"
Eta_pump=W_pump_u/W_shaft
```

| Head Loss, $h_L$, m | Pumping power $W_{pump,\,u}$ | Efficiency $\eta_{pump}$ |
|---|---|---|
| 0 | 6.13 | 0.613 |
| 1 | 6.38 | 0.638 |
| 2 | 6.62 | 0.662 |
| 3 | 6.87 | 0.687 |
| 4 | 7.11 | 0.711 |
| 5 | 7.36 | 0.736 |
| 6 | 7.60 | 0.760 |
| 7 | 7.85 | 0.785 |
| 8 | 8.09 | 0.809 |
| 9 | 8.34 | 0.834 |
| 10 | 8.58 | 0.858 |
| 11 | 8.83 | 0.883 |
| 12 | 9.07 | 0.907 |
| 13 | 9.32 | 0.932 |
| 14 | 9.56 | 0.956 |
| 15 | 9.81 | 0.981 |




***Discussion*** Note that the useful pumping power is used to raise the fluid and to overcome head losses. For a given power input, the pump that overcomes more head loss is more efficient.

**5-76**

**Solution** A pump with a specified shaft power and efficiency is used to raise water to a higher elevation. The maximum flow rate of water is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The elevation difference between the reservoirs is constant. **3** We assume the flow in the pipes to be *frictionless* since the *maximum* flow rate is to be determined, $\dot{E}_{\text{mech loss, piping}} = 0$.

***Properties*** We take the density of water to be $\rho = 1000$ kg/m$^3$.

***Analysis*** We choose points 1 and 2 at the free surfaces of the lower and upper reservoirs, respectively, and take the surface of the lower reservoir as the reference level ($z_1 = 0$). Both points are open to the atmosphere ($P_1 = P_2 = P_{atm}$) and the velocities at both locations are negligible ($V_1 = V_2 = 0$). Then the energy equation for steady incompressible flow through a control volume between these two points that includes the pump and the pipes reduces to

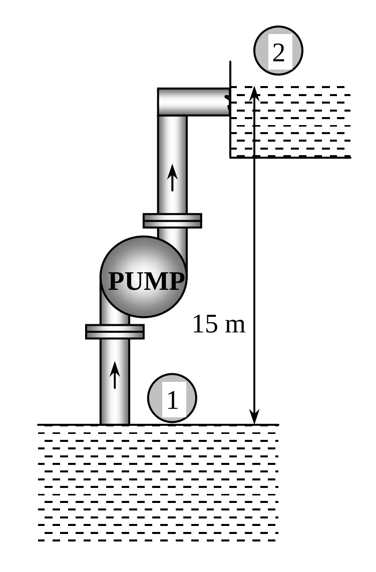
$$\dot{m}\left(\frac{P_1}{\rho}+\alpha_1\frac{V_1^2}{2}+gz_1\right)+\dot{W}_{\text{pump}} = \dot{m}\left(\frac{P_2}{\rho}+\alpha_2\frac{V_2^2}{2}+gz_2\right)+\dot{W}_{\text{turbine}}+\dot{E}_{\text{mech,loss}} \;\rightarrow\; \dot{W}_{\text{pump,u}} = \dot{m}gz_2 = \rho\dot{V}gz_2$$

since $\dot{E}_{\text{mech, loss}} = \dot{E}_{\text{mech loss, pump}}$ in this case and $\dot{W}_{\text{pump, u}} = \dot{W}_{\text{pump}} - \dot{E}_{\text{mech loss, pump}}$. The useful pumping power is

$$\dot{W}_{\text{pump,u}} = \eta_{\text{pump}}\dot{W}_{\text{pump, shaft}} = (0.82)(7\text{ hp}) = 5.74\text{ hp}$$

Substituting, the volume flow rate of water is determined to be

$$\dot{V} = \frac{\dot{W}_{\text{pump,u}}}{\rho g z_2} = \frac{5.74\text{ hp}}{(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(15\text{ m})}\left(\frac{745.7\text{ W}}{1\text{ hp}}\right)\left(\frac{1\text{ N}\cdot\text{m/s}}{1\text{ W}}\right)\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)$$
$$= \mathbf{0.0291\ m^3/s}$$




***Discussion*** This is the maximum flow rate since the frictional effects are ignored. In an actual system, the flow rate of water will be less because of friction in pipes.

**5-77**

**Solution** Water flows at a specified rate in a horizontal pipe whose diameter is decreased by a reducer. The pressures are measured before and after the reducer. The head loss in the reducer is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The pipe is horizontal. **3** The kinetic energy correction factors are given to be $\alpha_1 = \alpha_2 = \alpha = 1.05$.

***Properties*** We take the density of water to be $\rho = 1000$ kg/m$^3$.

***Analysis*** We take points 1 and 2 along the centerline of the pipe before and after the reducer, respectively. Noting that $z_1 = z_2$, the energy equation for steady incompressible flow through a control volume between these two points reduces to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad h_L = \frac{P_1 - P_2}{\rho g} + \frac{\alpha(V_1^2 - V_2^2)}{2g}$$

where

$$V_1 = \frac{\dot{V}}{A_1} = \frac{\dot{V}}{\pi D_1^2/4} = \frac{0.035 \text{ m}^3\text{/s}}{\pi(0.15 \text{ m})^2/4} = 1.98 \text{ m/s}$$

$$V_2 = \frac{\dot{V}}{A_2} = \frac{\dot{V}}{\pi D_2^2/4} = \frac{0.035 \text{ m}^3\text{/s}}{\pi(0.08 \text{ m})^2/4} = 6.96 \text{ m/s}$$



Substituting, the head loss in the reducer is determined to be

$$h_L = \frac{(470 - 440) \text{ kPa}}{(1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)}\left(\frac{1 \text{ kN/m}^2}{1 \text{ kPa}}\right)\left(\frac{1000 \text{ kg}\cdot\text{m/s}^2}{1 \text{ kN}}\right) + \frac{1.05[(1.98 \text{ m/s})^2 - (6.96 \text{ m/s})^2]}{2(9.81 \text{ m/s}^2)}$$
$$= 3.06 - 2.38 = \mathbf{0.68 \text{ m}}$$

***Discussion*** Note that the 0.79 m of the head loss is due to frictional effects and 2.27 m is due to the increase in velocity. This head loss corresponds to a power potential loss of

$$\dot{E}_{\text{mech loss, piping}} = \rho \dot{V} g h_L = (1000 \text{ kg/m}^3)(0.035 \text{ m}^3\text{/s})(9.81 \text{ m/s}^2)(0.79 \text{ m})\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ W}}{1 \text{ N}\cdot\text{m/s}}\right) = \mathbf{271 \text{ W}}$$

## 5-78

**Solution** A hose connected to the bottom of a tank is equipped with a nozzle at the end pointing straight up. The water is pressurized by a pump, and the height of the water jet is measured. The minimum pressure rise supplied by the pump is to be determined.

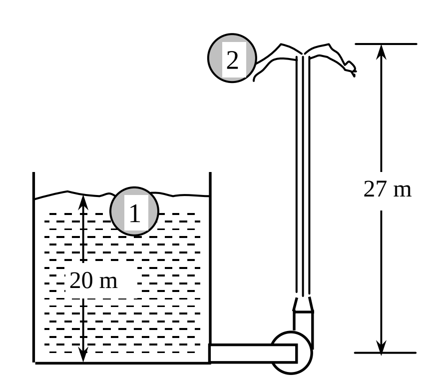
***Assumptions*** **1** The flow is steady and incompressible. **2** Friction between the water and air as well as friction in the hose is negligible. **3** The water surface is open to the atmosphere.

***Properties*** We take the density of water to be $\rho = 1000$ kg/m$^3$.

***Analysis*** We take point 1 at the free surface of water in the tank, and point 2 at the top of the water trajectory where $V_2 = 0$ and $P_1 = P_2 = P_{atm}$. Also, we take the reference level at the bottom of the tank. Noting that $z_1 = 20$ m and $z_2 = 27$ m, $h_L = 0$ (to get the minimum value for required pressure rise), and that the fluid velocity at the free surface of the tank is very low ($V_1 \cong 0$), the energy equation for steady incompressible flow through a control volume between these two points that includes the pump and the water stream reduces to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L$$

$$\rightarrow \quad h_{\text{pump, u}} = z_2 - z_1$$




Substituting,

$$h_{\text{pump, u}} = 27 - 20 = 7 \text{ m}$$

A water column height of 7 m corresponds to a pressure rise of

$$\Delta P_{\text{pump, min}} = \rho g h_{\text{pump, u}} = (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(7 \text{ m})\left(\frac{1 \text{ N}}{1000 \text{ kg}\cdot\text{m/s}^2}\right)$$

$$= 68.7 \text{ kN/m}^2 = \mathbf{68.7\ kPa}$$

Therefore, the pump must supply a minimum pressure rise of 68.7 kPa.

***Discussion*** The result obtained above represents the minimum value, and should be interpreted accordingly. In reality, a larger pressure rise will need to be supplied to overcome friction.

---

## 5-79

**Solution** The available head of a hydraulic turbine and its overall efficiency are given. The electric power output of this turbine is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The available head remains constant.

***Properties*** We take the density of water to be $\rho = 1000$ kg/m$^3$.

***Analysis*** When the turbine head is available, the corresponding power output is determined from

Eff.=78%

Turbine Generator

$$\dot{W}_{\text{turbine}} = \eta_{\text{turbine}} \dot{m} g h_{\text{turbine}} = \eta_{\text{turbine}} \rho \dot{V} g h_{\text{turbine}}$$

Substituting,

$$\dot{W}_{\text{turbine}} = 0.78(1000 \text{ kg/m}^3)(0.25 \text{ m}^3/\text{s})(9.81 \text{ m/s}^2)(85 \text{ m})\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ kW}}{1000 \text{ N}\cdot\text{m/s}}\right) = \mathbf{163\ kW}$$

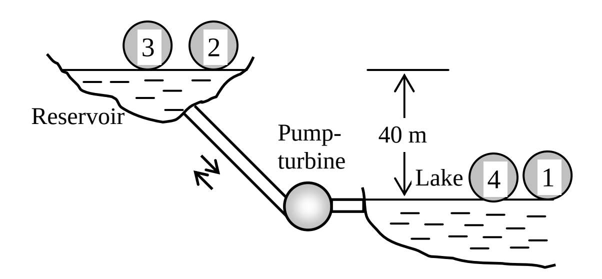
***Discussion*** The power output of a hydraulic turbine is proportional to the available turbine head and the flow rate.

---

**5-80**

**Solution** An entrepreneur is to build a large reservoir above the lake level, and pump water from the lake to the reservoir at night using cheap power, and let the water flow from the reservoir back to the lake during the day, producing power. The potential revenue this system can generate per year is to be determined.

***Assumptions*** **1** The flow in each direction is steady and incompressible. **2** The elevation difference between the lake and the reservoir can be taken to be constant, and the elevation change of reservoir during charging and discharging is disregarded. **3** The given unit prices remain constant. **4** The system operates every day of the year for 10 hours in each mode.




***Properties*** We take the density of water to be $\rho$ = 1000 kg/m$^3$.

***Analysis*** We choose points 1 and 2 at the free surfaces of the lake and the reservoir, respectively, and take the surface of the lake as the reference level. Both points are open to the atmosphere ($P_1 = P_2 = P_{atm}$) and the velocities at both locations are negligible ($V_1 = V_2 = 0$). Then the energy equation in terms of heads for steady incompressible flow through a control volume between these two points that includes the pump (or the turbine) and the pipes reduces to

*Pump mode*: $$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \rightarrow$$

$$h_{\text{pump, u}}=z_2+h_L=40+4=44\text{ m}$$

*Turbine mode*: (switch points 1 and 2 so that 1 is on inlet side) $\rightarrow$ $h_{\text{turbine, e}}=z_1-h_L=40-4=36\text{ m}$

The pump and turbine power corresponding to these heads are

$$\dot{W}_{\text{pump, elect}}=\frac{\dot{W}_{\text{pump, u}}}{\eta_{\text{pump-motor}}}=\frac{\rho\dot{V}gh_{\text{pump, u}}}{\eta_{\text{pump-motor}}}$$
$$=\frac{(1000\text{ kg/m}^3)(2\text{ m}^3\text{/s})(9.81\text{ m/s}^2)(44\text{ m})}{0.75}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kW}}{1000\text{ N}\cdot\text{m/s}}\right)=1151\text{ kW}$$

$$\dot{W}_{\text{turbine}}=\eta_{\text{turbine-gen}}\dot{m}gh_{\text{turbine, e}}=\eta_{\text{turbine-gen}}\rho\dot{V}gh_{\text{turbine, e}}$$
$$=0.75(1000\text{ kg/m}^3)(2\text{ m}^3\text{/s})(9.81\text{ m/s}^2)(36\text{ m})\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kW}}{1000\text{ N}\cdot\text{m/s}}\right)=530\text{ kW}$$

Then the power cost of the pump, the revenue generated by the turbine, and the net income (revenue minus cost) per year become

$$\text{Cost}=\dot{W}_{\text{pump, elect}}\Delta t\times\text{Unit price}=(1151\text{ kW})(365\times 10\text{ h/year})(\$0.03\text{/kWh})=\$126{,}035\text{/year}$$

$$\text{Revenue}=\dot{W}_{\text{turbine}}\Delta t\times\text{Unit price}=(530\text{ kW})(365\times 10\text{ h/year})(\$0.08\text{/kWh})=\$154{,}760\text{/year}$$

Net income = Revenue – Cost = 154,760 –126,035 = **$28,725/year ≅ $28,700/year**

***Discussion*** It appears that this pump-turbine system has a potential annual income of about $29,000. A decision on such a system will depend on the initial cost of the system, its life, the operating and maintenance costs, the interest rate, and the length of the contract period, among other things.

**5-81**

**Solution** Water flows through a horizontal pipe at a specified rate. The pressure drop across a valve in the pipe is measured. The corresponding head loss and the power needed to overcome it are to be determined.

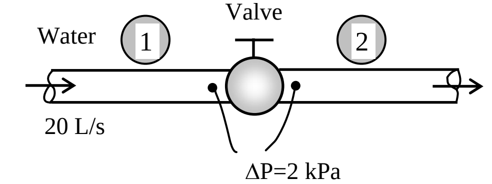
***Assumptions*** **1** The flow is steady and incompressible. **2** The pipe is given to be horizontal (otherwise the elevation difference across the valve is negligible). **3** The mean flow velocities at the inlet and the exit of the valve are equal since the pipe diameter is constant.

***Properties*** We take the density of water to be $\rho = 1000$ kg/m$^3$.

***Analysis*** We take the valve as the control volume, and points 1 and 2 at the inlet and exit of the valve, respectively. Noting that $z_1 = z_2$ and $V_1 = V_2$, the energy equation for steady incompressible flow through this control volume reduces to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad h_L=\frac{P_1-P_2}{\rho g}$$

Substituting,




$$h_L=\frac{2\text{ kN/m}^2}{(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)}\left(\frac{1000\text{ kg}\cdot\text{m/s}^2}{1\text{ kN}}\right)=\mathbf{0.204\ m}$$

The useful pumping power needed to overcome this head loss is

$$\dot W_{\text{pump, u}}=\dot m g h_L=\rho\dot V g h_L$$

$$=(1000\text{ kg/m}^3)(0.020\text{ m}^3\text{/s})(9.81\text{ m/s}^2)(0.204\text{ m})\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ W}}{1\text{ N}\cdot\text{m/s}}\right)=\mathbf{40\ W}$$

Therefore, this valve would cause a head loss of 0.204 m, and it would take 40 W of useful pumping power to overcome it.

***Discussion*** The required useful pumping power could also be determined from

$$\dot W_{\text{pump}}=\dot V\Delta P=(0.020\text{ m}^3\text{/s})(2000\text{ Pa})\left(\frac{1\text{ W}}{1\text{ Pa}\cdot\text{m}^3\text{/s}}\right)=\mathbf{40\ W}$$

## 5-82E

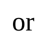
**Solution** A hose connected to the bottom of a pressurized tank is equipped with a nozzle at the end pointing straight up. The minimum tank air pressure (gage) corresponding to a given height of water jet is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** Friction between water and air as well as friction in the hose is negligible. **3** The water surface is open to the atmosphere.

***Properties*** We take the density of water to be $\rho = 62.4$ lbm/ft$^3$.

***Analysis*** We take point 1 at the free surface of water in the tank, and point 2 at the top of the water trajectory where $V_2 = 0$ and $P_1 = P_2 = P_{atm}$. Also, we take the reference level at the bottom of the tank. Noting that $z_1 = 66$ ft and $z_2 = 90$ ft, $h_L = 0$ (to get the minimum value for the required air pressure), and that the fluid velocity at the free surface of the tank is very low ($V_1 \cong 0$), the energy equation for steady incompressible flow through a control volume between these two points reduces to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L$$


or

$$\frac{P_1-P_{atm}}{\rho g}=z_2-z_1 \rightarrow \frac{P_{1,gage}}{\rho g}=z_2-z_1$$

Rearranging and substituting, the gage pressure of pressurized air in the tank is determined to be

$$P_{1,\text{ gage}}=\rho g(z_2-z_1)=(62.4\text{ lbm/ft}^3)(32.2\text{ ft/s}^2)(90-66\text{ ft})\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1\text{ psi}}{144\text{ lbf/ft}^2}\right)=\mathbf{10.4\text{ psi}}$$

Therefore, the gage air pressure on top of the water tank must be at least 10.4 psi.

***Discussion*** The result obtained above represents the minimum value, and should be interpreted accordingly. In reality, a larger pressure will be needed to overcome friction.

---

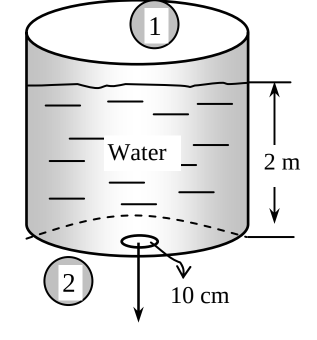
## 5-83

**Solution** A water tank open to the atmosphere is initially filled with water. A sharp-edged orifice at the bottom drains to the atmosphere. The initial discharge velocity from the tank is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The tank is open to the atmosphere. **3** The kinetic energy correction factor at the orifice is given to be $\alpha_2 = \alpha = 1.2$.

***Analysis*** We take point 1 at the free surface of the tank, and point 2 at the exit of the orifice. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface of the tank is very low ($V_1 \cong 0$), the energy equation between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L$$




which yields

$$z_1+\alpha_2\frac{V_2^2}{2g}=z_2+h_L$$

Solving for $V_2$ and substituting,

$$V_2=\sqrt{2g(z_1-z_2-h_L)/\alpha}=\sqrt{2(9.81\text{ m/s}^2)(2-0.3\text{ m})/1.2}=\mathbf{5.27\text{ m/s}}$$

***Discussion*** This is the velocity that will prevail at the beginning. The mean flow velocity will decrease as the water level in the tank decreases.

---

**5-84**

**Solution** Water enters a hydraulic turbine-generator system with a known flow rate, pressure drop, and efficiency. The net electric power output is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** All losses in the turbine are accounted for by turbine efficiency and thus $h_L = 0$. **3** The elevation difference across the turbine is negligible. **4** The effect of the kinetic energy correction factors is negligible, $\alpha_1 = \alpha_2 = \alpha = 1$.

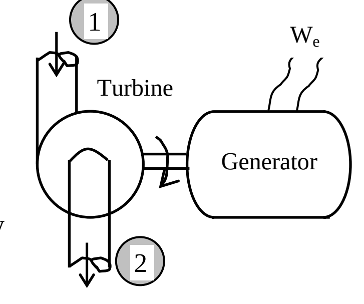
***Properties*** We take the density of water to be 1000 kg/m$^3$ and the density of mercury to be 13,560 kg/m$^3$.

***Analysis*** We choose points 1 and 2 at the inlet and the exit of the turbine, respectively. Noting that the elevation effects are negligible, the energy equation in terms of heads for the turbine reduces to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad h_{\text{turbine, e}}=\frac{P_1-P_2}{\rho_{\text{water}}g}+\frac{\alpha(V_1^2-V_2^2)}{2g} \qquad (1)$$

where

$$V_1=\frac{\dot V}{A_1}=\frac{\dot V}{\pi D_1^2/4}\frac{0.6\ \text{m}^3/\text{s}}{\pi(0.30\ \text{m})^2/4}=8.49\ \text{m/s}$$




$$V_2=\frac{\dot V}{A_2}=\frac{\dot V}{\pi D_2^2/4}=\frac{0.6\ \text{m}^3/\text{s}}{\pi(0.25\ \text{m})^2/4}=12.2\ \text{m/s}$$

The pressure drop corresponding to a differential height of 1.2 m in the mercury manometer is

$$P_1-P_2=(\rho_{\text{Hg}}-\rho_{\text{water}})gh$$
$$=[(13{,}560-1000)\ \text{kg/m}^3](9.81\ \text{m/s}^2)(1.2\ \text{m})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)$$
$$=148\ \text{kN/m}^2=148\ \text{kPa}$$

Substituting into Eq. (1), the turbine head is determined to be

$$h_{\text{turbine, e}}=\frac{148\ \text{kN/m}^2}{(1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)}\left(\frac{1000\ \text{kg}\cdot\text{m/s}^2}{1\ \text{kN}}\right)+(1.0)\frac{(8.49\ \text{m/s})^2-(12.2\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)}=15.1-3.9=11.2\ \text{m}$$

Then the net electric power output of this hydroelectric turbine becomes

$$\dot W_{\text{turbine}}=\eta_{\text{turbine-gen}}\dot m g h_{\text{turbine, e}}=\eta_{\text{turbine-gen}}\rho\dot V g h_{\text{turbine, e}}$$
$$=0.83(1000\ \text{kg/m}^3)(0.6\ \text{m}^3/\text{s})(9.81\ \text{m/s}^2)(11.2\ \text{m})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kW}}{1000\ \text{N}\cdot\text{m/s}}\right)=\mathbf{55\ kW}$$

***Discussion*** It appears that this hydroelectric turbine will generate 55 kW of electric power under given conditions. Note that almost half of the available pressure head is discarded as kinetic energy. This demonstrates the need for a larger turbine exit area and better recovery. For example, the power output can be increased to 74 kW by redesigning the turbine and making the exit diameter of the pipe equal to the inlet diameter, $D_2 = D_1$. Further, if a much larger exit diameter is used and the exit velocity is reduced to a very low level, the power generation can increase to as much as 92 kW.

**5-85**

**Solution** The velocity profile for turbulent flow in a circular pipe is given. The kinetic energy correction factor for this flow is to be determined.

***Analysis*** The velocity profile is given by $u(r) = u_{\max}(1 - r/R)^{1/n}$ with $n$ = 7 The kinetic energy correction factor is then expressed as

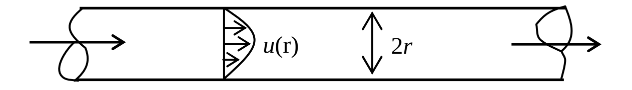
$$\alpha = \frac{1}{A}\int_A \left(\frac{u(r)}{V_{avg}}\right)^3 dA = \frac{1}{AV_{avg}^3}\int_A u(r)^3\, dA = \frac{1}{\pi R^2 V_{avg}^3}\int_{r=0}^{R} u_{\max}^3 \left(1 - \frac{r}{R}\right)^{\frac{3}{n}} (2\pi r)dr = \frac{2u_{\max}^3}{R^2 V_{avg}^3}\int_{r=0}^{R}\left(1 - \frac{r}{R}\right)^{\frac{3}{n}} rdr$$

where the average velocity is

$$V_{avg} = \frac{1}{A}\int_A u(r)dA = \frac{1}{\pi R^2}\int_{r=0}^{R} u_{\max}\left(1 - \frac{r}{R}\right)^{1/n}(2\pi r)dr = \frac{2u_{\max}}{R^2}\int_{r=0}^{R}\left(1 - \frac{r}{R}\right)^{1/n} rdr$$

From integral tables,

$$\int (a + bx)^n\, xdx = \frac{(a + bx)^{n+2}}{b^2(n+2)} - \frac{a(a + bx)^{n+1}}{b^2(n+1)}$$




Then,

$$\int_{r=0}^{R} u(r)rdr = \int_{r=0}^{R}\left(1 - \frac{r}{R}\right)^{1/n} rdr = \left.\frac{(1 - r/R)^{\frac{1}{n}+2}}{\frac{1}{R^2}(\frac{1}{n}+2)} - \frac{(1 - r/R)^{\frac{1}{n}+1}}{\frac{1}{R^2}(\frac{1}{n}+1)}\right|_{r=0}^{R} = \frac{n^2 R^2}{(n+1)(2n+1)}$$

$$\int_{r=0}^{R} u(r)^3 rdr = \int_{r=0}^{R}\left(1 - \frac{r}{R}\right)^{3/n} rdr = \left.\frac{(1 - r/R)^{\frac{3}{n}+2}}{\frac{1}{R^2}(\frac{3}{n}+2)} - \frac{(1 - r/R)^{\frac{3}{n}+1}}{\frac{1}{R^2}(\frac{3}{n}+1)}\right|_{r=0}^{R} = \frac{n^2 R^2}{(n+3)(2n+3)}$$

Substituting,

$$V_{avg} = \frac{2u_{\max}}{R^2}\frac{n^2R^2}{(n+1)(2n+1)} = \frac{2n^2 u_{\max}}{(n+1)(2n+1)} = 0.8167 u_{\max}$$

and

$$\alpha = \frac{2u_{\max}^3}{R^2}\left(\frac{2n^2 u_{\max}}{(n+1)(2n+1)}\right)^{-3}\frac{n^2R^2}{(n+3)(2n+3)} = \frac{(n+1)^3(2n+1)^3}{4n^4(n+3)(2n+3)} = \frac{(7+1)^3(2\times 7+1)^3}{4\times 7^4(7+3)(2\times 7+3)} = \mathbf{1.06}$$

***Discussion*** Note that ignoring the kinetic energy correction factor results in an error of just 6% in this case in the kinetic energy term (which may be small itself). Considering that the uncertainties in some terms are usually more that 6%, we can usually ignore this correction factor in turbulent pipe flow analyses. However, for laminar pipe flow analyses, $\alpha$ is equal to 2.0 for fully developed laminar pipe flow, and ignoring $\alpha$ may lead to significant errors.

---

**5-86**

**Solution** A pump is pumping oil at a specified rate. The pressure rise of oil in the pump is measured, and the motor efficiency is specified. The mechanical efficiency of the pump is to be determined.

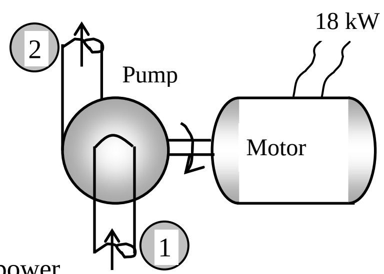
***Assumptions*** **1** The flow is steady and incompressible. **2** The elevation difference across the pump is negligible. **3** All the losses in the pump are accounted for by the pump efficiency and thus $h_L = 0$. **4** The kinetic energy correction factors are given to be $\alpha_1 = \alpha_2 = \alpha = 1.05$.

***Properties*** The density of oil is given to be $\rho = 860$ kg/m$^3$.

***Analysis*** We take points 1 and 2 at the inlet and the exit of the pump, respectively. Noting that $z_1 = z_2$, the energy equation for the pump reduces to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad h_{\text{pump, u}}=\frac{P_2-P_1}{\rho g}+\frac{\alpha(V_2^2-V_1^2)}{2g}$$

where




$$V_1=\frac{\dot{V}}{A_1}=\frac{\dot{V}}{\pi D_1^2/4}=\frac{0.1\,\text{m}^3/\text{s}}{\pi(0.08\,\text{m})^2/4}=19.9\text{ m/s}$$

$$V_2=\frac{\dot{V}}{A_2}=\frac{\dot{V}}{\pi D_2^2/4}=\frac{0.1\,\text{m}^3/\text{s}}{\pi(0.12\,\text{m})^2/4}=8.84\text{ m/s}$$

Substituting, the useful pump head and the corresponding useful pumping power are determined to be

$$h_{\text{pump, u}}=\frac{400{,}000\text{ N/m}^2}{(860\text{ kg/m}^3)(9.81\text{ m/s}^2)}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)+\frac{1.05[(8.84\text{ m/s})^2-(19.9\text{ m/s})^2]}{2(9.81\text{ m/s}^2)}=47.4-17.0=30.4\text{ m}$$

$$\dot{W}_{\text{pump,u}}=\rho\dot{V}gh_{\text{pump, u}}=(860\text{ kg/m}^3)(0.1\text{ m}^3/\text{s})(9.81\text{ m/s}^2)(30.4\text{ m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}}\right)=25.6\text{ kW}$$

Then the shaft pumping power and the mechanical efficiency of the pump become

$$\dot{W}_{\text{pump,shaft}}=\eta_{\text{motor}}\dot{W}_{\text{electric}}=(0.90)(35\text{ kW})=31.5\text{ kW}$$

$$\eta_{\text{pump}}=\frac{\dot{W}_{\text{pump, u}}}{\dot{W}_{\text{pump, shaft}}}=\frac{25.6\text{ kW}}{31.5\text{ kW}}=0.813=\mathbf{81.3\%}$$

***Discussion*** The overall efficiency of this pump/motor unit is the product of the mechanical and motor efficiencies, which is 0.9 × 0.813 = 0.73.

---

**5-87E**

**Solution** Water is pumped from a lake to a nearby pool by a pump with specified power and efficiency. The head loss of the piping system and the mechanical power used to overcome it are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The elevation difference between the lake and the free surface of the pool is constant. **3** All the losses in the pump are accounted for by the pump efficiency and thus $h_L$ represents the losses in piping.

***Properties*** We take the density of water to be $\rho = 62.4$ lbm/ft$^3$.

***Analysis*** The useful pumping power and the corresponding useful pumping head are

$$\dot{W}_{\text{pump,u}} = \eta_{\text{pump}}\dot{W}_{\text{pump}} = (0.73)(12\text{ hp}) = 8.76\text{ hp}$$

$$h_{\text{pump, u}} = \frac{\dot{W}_{\text{pump, u}}}{\dot{m}g} = \frac{\dot{W}_{\text{pump, u}}}{\rho\dot{V}g}$$

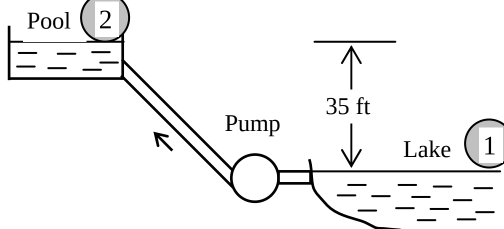
$$= \frac{8.76\text{ hp}}{(62.4\text{ lbm/ft}^3)(1.2\text{ ft}^3/\text{s})(32.2\text{ ft/s}^2)}\left(\frac{32.2\text{ lbm}\cdot\text{ft/s}^2}{1\text{ lbf}}\right)\left(\frac{550\text{ lbf}\cdot\text{ft/s}}{1\text{ hp}}\right) = 64.3\text{ ft}$$

We choose points 1 and 2 at the free surfaces of the lake and the pool, respectively. Both points are open to the atmosphere ($P_1 = P_2 = P_{atm}$) and the velocities at both locations are negligible ($V_1 = V_2 = 0$). Then the energy equation for steady incompressible flow through a control volume between these two points that includes the pump and the pipes reduces to

$$\frac{P_1}{\rho g} + \alpha_1\frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2\frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \to \quad h_L = h_{\text{pump, u}} + z_1 - z_2$$

Substituting, the head loss is determined to be

$$h_L = h_{\text{pump, u}} - (z_2 - z_1) = 64.3 - 35 = \mathbf{29.3\text{ ft}}$$




Then the power used to overcome it becomes

$$\dot{E}_{\text{mech loss, piping}} = \rho\dot{V}gh_L$$

$$= (62.4\text{ lbm/ft}^3)(1.2\text{ ft}^3/\text{s})(32.2\text{ ft/s}^2)(29.3\text{ ft})\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1\text{ hp}}{550\text{ lbf}\cdot\text{ft/s}}\right)$$

$$= \mathbf{4.0\text{ hp}}$$

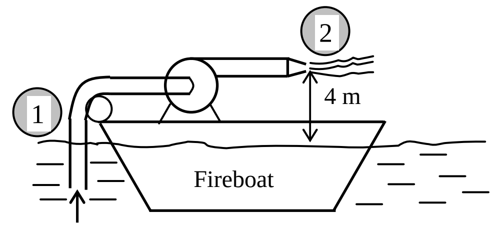
***Discussion*** Note that the pump must raise the water an additional height of 29.3 ft to overcome the frictional losses in pipes, which requires an additional useful pumping power of about 4 hp.

**5-88**

**Solution** A fireboat is fighting fires by drawing sea water and discharging it through a nozzle. The head loss of the system and the elevation of the nozzle are given. The shaft power input to the pump and the water discharge velocity are to be determined.

***Assumptions*** 1 The flow is steady and incompressible. **2** The effect of the kinetic energy correction factors is negligible, $\alpha = 1$.

***Properties*** The density of sea water is given to be $\rho$ =1030 kg/m$^3$.




***Analysis*** We take point 1 at the free surface of the sea and point 2 at the nozzle exit. Noting that $P_1 = P_2 = P_{atm}$ and $V_1 \cong 0$ (point 1 is at the free surface; not at the pipe inlet), the energy equation for the control volume between 1 and 2 that includes the pump and the piping system reduces to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad h_{\text{pump, u}} = z_2 - z_1 + \alpha_2 \frac{V_2^2}{2g} + h_L$$

where the water discharge velocity is

$$V_2 = \frac{\dot{V}}{A_2} = \frac{\dot{V}}{\pi D_2^2 / 4} = \frac{0.1 \text{ m}^3/\text{s}}{\pi (0.05 \text{ m})^2 / 4} = 50.93 \text{ m/s} \cong \mathbf{50.9 \text{ m/s}}$$

Substituting, the useful pump head and the corresponding useful pump power are determined to be

$$h_{\text{pump, u}} = (4 \text{ m}) + (1)\frac{(50.93 \text{ m/s})^2}{2(9.81 \text{ m/s}^2)} + (3 \text{ m}) = 139.2 \text{ m}$$

$$\dot{W}_{\text{pump,u}} = \rho \dot{V} g h_{\text{pump, u}} = (1030 \text{ kg/m}^3)(0.1 \text{ m}^3/\text{s})(9.81 \text{ m/s}^2)(139.2 \text{ m})\left(\frac{1 \text{ kN}}{1000 \text{ kg} \cdot \text{m/s}^2}\right)\left(\frac{1 \text{ kW}}{1 \text{ kN} \cdot \text{m/s}}\right)$$

$$= 140.7 \text{ kW}$$

Then the required shaft power input to the pump becomes

$$\dot{W}_{\text{pump, shaft}} = \frac{\dot{W}_{\text{pump, u}}}{\eta_{\text{pump}}} = \frac{140.7 \text{ kW}}{0.70} = \mathbf{201 \text{ kW}}$$

***Discussion*** Note that the pump power is used primarily to increase the kinetic energy of water.

## Review Problems

**5-89**

**Solution** A water tank open to the atmosphere is initially filled with water. The tank discharges to the atmosphere through a long pipe connected to a valve. The initial discharge velocity from the tank and the time required to empty the tank are to be determined.

***Assumptions*** **1** The flow is incompressible. **2** The draining pipe is horizontal. **3** The tank is considered to be empty when the water level drops to the center of the valve.

***Analysis*** (*a*) Substituting the known quantities, the discharge velocity can be expressed as

$$V=\sqrt{\frac{2gz}{1.5+fL/D}}=\sqrt{\frac{2gz}{1.5+0.015(100\text{ m})/(0.10\text{ m})}}=\sqrt{0.1212gz}$$

Then the initial discharge velocity becomes

$$V_1=\sqrt{0.1212gz_1}=\sqrt{0.1212(9.81\text{ m/s}^2)(2\text{ m})}=\mathbf{1.54\ m/s}$$

where $z$ is the water height relative to the center of the orifice at that time.

(*b*) The flow rate of water from the tank can be obtained by multiplying the discharge velocity by the pipe cross-sectional area,

$$\dot{V}=A_{\text{pipe}}V_2=\frac{\pi D^2}{4}\sqrt{0.1212gz}$$

Then the amount of water that flows through the pipe during a differential time interval $dt$ is

$$dV=\dot{V}dt=\frac{\pi D^2}{4}\sqrt{0.1212gz}\,dt \qquad (1)$$

which, from conservation of mass, must be equal to the decrease in the volume of water in the tank,

$$dV=A_{\tan k}(-dz)=-\frac{\pi D_0^2}{4}dz \qquad (2)$$

where $dz$ is the change in the water level in the tank during $dt$. (Note that $dz$ is a negative quantity since the positive direction of $z$ is upwards. Therefore, we used $-dz$ to get a positive quantity for the amount of water discharged). Setting Eqs. (1) and (2) equal to each other and rearranging,

$$\frac{\pi D^2}{4}\sqrt{0.1212gz}\,dt=-\frac{\pi D_0^2}{4}dz \rightarrow dt=-\frac{D_0^2}{D^2}\frac{dz}{\sqrt{0.1212gz}}=-\frac{D_0^2}{D^2\sqrt{0.1212g}}z^{-\frac{1}{2}}dz$$

The last relation can be integrated easily since the variables are separated. Letting $t_f$ be the discharge time and integrating it from $t = 0$ when $z = z_1$ to $t = t_f$ when $z = 0$ (completely drained tank) gives

$$\int_{t=0}^{t_f}dt=-\frac{D_0^2}{D^2\sqrt{0.1212g}}\int_{z=z_1}^{0}z^{-1/2}dz \rightarrow t_f=-\frac{D_0^2}{D^2\sqrt{0.1212g}}\left|\frac{z^{\frac{1}{2}}}{\frac{1}{2}}\right|_{z_1}^{0}=\frac{2D_0^2}{D^2\sqrt{0.1212g}}z_1^{\frac{1}{2}}$$

Simplifying and substituting the values given, the draining time is determined to be

$$t_f=\frac{2D_0^2}{D^2}\sqrt{\frac{z_1}{0.1212g}}=\frac{2(10\text{ m})^2}{(0.1\text{ m})^2}\sqrt{\frac{2\text{ m}}{0.1212(9.81\text{ m/s}^2)}}=25{,}940\text{ s}=\mathbf{7.21\ h}$$

***Discussion*** The draining time can be shortened considerably by installing a pump in the pipe.

**5-90**

**Solution** The rate of accumulation of water in a pool and the rate of discharge are given. The rate supply of water to the pool is to be determined.

***Assumptions*** **1** Water is supplied and discharged steadily. **2** The rate of evaporation of water is negligible. **3** No water is supplied or removed through other means.

***Analysis*** The conservation of mass principle applied to the pool requires that the rate of increase in the amount of water in the pool be equal to the difference between the rate of supply of water and the rate of discharge. That is,

$$\frac{dm_{pool}}{dt} = \dot{m}_i - \dot{m}_e \quad \rightarrow \quad \dot{m}_i = \frac{dm_{pool}}{dt} + \dot{m}_e \quad \rightarrow \quad \dot{V}_i = \frac{d\dot{V}_{pool}}{dt} + \dot{V}_e$$

since the density of water is constant and thus the conservation of mass is equivalent to conservation of volume. The rate of discharge of water is

$$\dot{V}_e = A_e V_e = (\pi D^2/4)V_e = [\pi(0.05\text{ m})^2/4](5\text{ m/s}) = 0.00982\text{ m}^3/\text{s}$$

The rate of accumulation of water in the pool is equal to the cross-section of the pool times the rate at which the water level rises,

$$\frac{dV_{pool}}{dt} = A_{\text{cross-section}}V_{\text{level}} = (3\text{ m}\times 4\text{ m})(0.015\text{ m/min}) = 0.18\text{ m}^3/\text{min} = 0.00300\text{ m}^3/\text{s}$$

Substituting, the rate at which water is supplied to the pool is determined to be

$$\dot{V}_i = \frac{dV_{pool}}{dt} + \dot{V}_e = 0.003 + 0.00982 = 0.01282\text{ m}^3/\text{s} \cong \mathbf{0.0128\ m^3/s}$$

Therefore, water is supplied at a rate of 0.01282 $m^3$/s = 12.82 L/s.

***Discussion*** This is a very simple application of the conservation of mass equations.

**5-91**

**Solution** A fluid is flowing in a circular pipe. A relation is to be obtained for the average fluid velocity in terms of *V*(*r*), *R*, and *r*.

***Analysis*** Choosing a circular ring of area $dA = 2\pi r dr$ as our differential area, the mass flow rate through a cross-sectional area can be expressed as

$$\dot{m} = \int_A \rho V(r)dA = \int_0^R \rho V(r) 2\pi r\, dr$$

Setting this equal to and solving for $V_{avg}$,

$$V_{avg} = \frac{2}{R^2}\int_0^R V(r)r\, dr$$

***Discussion*** If *V* were a function of both *r* and *θ*, we would also need to integrate with respect to *θ*.

**5-92**

**Solution** Air is accelerated in a nozzle. The density of air at the nozzle exit is to be determined.

***Assumptions*** Flow through the nozzle is steady.

***Properties*** The density of air is given to be 4.18 kg/m³ at the inlet.

***Analysis*** There is only one inlet and one exit, and thus $\dot{m}_1 = \dot{m}_2 = \dot{m}$. Then,

$$\dot{m}_1 = \dot{m}_2$$
$$\rho_1 A_1 V_1 = \rho_2 A_2 V_2$$
$$\rho_2 = \frac{A_1}{A_2}\frac{V_1}{V_2}\rho_1 = 2\frac{120\text{ m/s}}{380\text{ m/s}}(4.18\text{ kg/m}^3) = \mathbf{2.64\ kg/m^3}$$

1 AIR 2

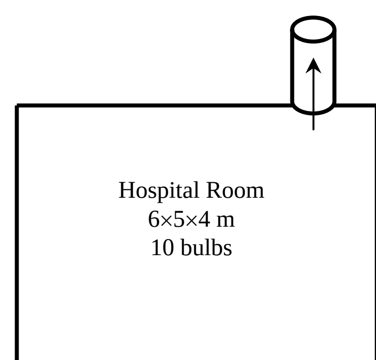
***Discussion*** Note that the density of air decreases considerably despite a decrease in the cross-sectional area of the nozzle.

**5-93**

**Solution** The air in a hospital room is to be replaced every 20 minutes. The minimum diameter of the duct is to be determined if the air velocity is not to exceed a certain value.

***Assumptions*** **1** The volume occupied by the furniture etc in the room is negligible. **2** The incoming conditioned air does not mix with the air in the room.

***Analysis*** The volume of the room is




$$\mathcal{V} = (6\text{ m})(5\text{ m})(4\text{ m}) = 120\text{ m}^3$$

To empty this air in 20 min, the volume flow rate must be

$$\dot{\mathcal{V}} = \frac{\mathcal{V}}{\Delta t} = \frac{120\text{ m}^3}{20\times 60\text{ s}} = 0.10\text{ m}^3/\text{s}$$

If the mean velocity is 5 m/s, the diameter of the duct is

$$\dot{\mathcal{V}} = AV = \frac{\pi D^2}{4}V \quad\rightarrow\quad D = \sqrt{\frac{4\dot{\mathcal{V}}}{\pi V}} = \sqrt{\frac{4(0.10\text{ m}^3/\text{s})}{\pi(5\text{ m/s})}} = \mathbf{0.16\ m}$$

Therefore, the diameter of the duct must be at least 0.16 m to ensure that the air in the room is exchanged completely within 20 min while the mean velocity does not exceed 5 m/s.

***Discussion*** This problem shows that engineering systems are sized to satisfy certain constraints imposed by certain considerations.

**5-94**

**Solution** Water discharges from the orifice at the bottom of a pressurized tank. The time it will take for half of the water in the tank to be discharged and the water level after 10 s are to be determined.

***Assumptions*** **1** The flow is incompressible, and the frictional effects are negligible. **2** The tank air pressure above the water level is maintained constant.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** We take point 1 at the free surface of the tank, and point 2 at the exit of orifice. We take the positive direction of $z$ to be upwards with reference level at the orifice ($z_2 = 0$). Fluid at point 2 is open to the atmosphere (and thus $P_2 = P_{atm}$) and the velocity at the free surface is very low ($V_1 \cong 0$). Then,

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \quad \rightarrow \quad \frac{P_1}{\rho g} + z_1 = \frac{P_{atm}}{\rho g} + \frac{V_2^2}{2g} \quad \rightarrow \quad V_2 = \sqrt{2gz_1 + 2P_{1,\text{gage}}/\rho}$$

or, $V_2 = \sqrt{2gz + 2P_{1,\text{gage}}/\rho}$ where $z$ is the water height in the tank at any time $t$. Water surface moves down as the tank drains, and the value of $z$ changes from $H$ initially to $0$ when the tank is emptied completely.

We denote the diameter of the orifice by $D$, and the diameter of the tank by $D_o$. The flow rate of water from the tank is obtained by multiplying the discharge velocity by the orifice cross-sectional area,

$$\dot{V} = A_{\text{orifice}} V_2 = \frac{\pi D^2}{4}\sqrt{2gz + 2P_{1,\text{gage}}/\rho}$$

Then the amount of water that flows through the orifice during a differential time interval $dt$ is

$$dV = \dot{V}dt = \frac{\pi D^2}{4}\sqrt{2gz + 2P_{1,\text{gage}}/\rho}\,dt \qquad (1)$$

which, from conservation of mass, must be equal to the decrease in the volume of water in the tank,

$$dV = A_{\text{tank}}(-dz) = -\frac{\pi D_0^2}{4}dz \qquad (2)$$

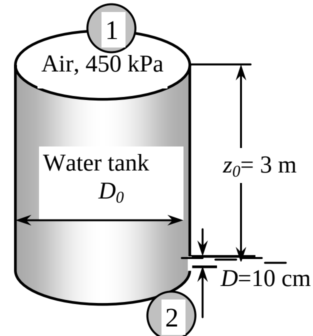
where $dz$ is the change in the water level in the tank during $dt$. (Note that $dz$ is a negative quantity since the positive direction of $z$ is upwards. Therefore, we used $-dz$ to get a positive quantity for the amount of water discharged). Setting Eqs. (1) and (2) equal to each other and rearranging,

$$\frac{\pi D^2}{4}\sqrt{2gz + 2P_{1,\text{gage}}/\rho}\,dt = -\frac{\pi D_0^2}{4}dz \quad \rightarrow \quad dt = -\frac{D_0^2}{D^2}\sqrt{\frac{1}{2gz + 2P_{1,\text{gage}}/\rho}}\,dz$$

The last relation can be integrated since the variables are separated. Letting $t_f$ be the discharge time and integrating it from $t = 0$ when $z = z_0$ to $t = t$ when $z = z$ gives

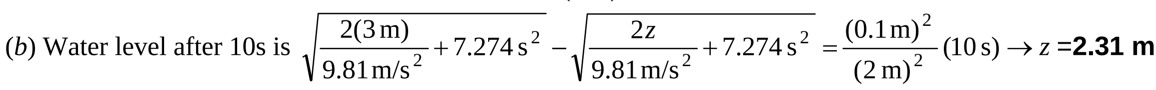
$$\sqrt{\frac{2z_0}{g} + \frac{2P_{1,gage}}{\rho g^2}} - \sqrt{\frac{2z}{g} + \frac{2P_{1,gage}}{\rho g^2}} = \frac{D_0^2}{D^2}t$$

where $$\frac{2P_{1,\text{gage}}}{\rho g^2} = \frac{2(450-100)\ \text{kN/m}^2}{(1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)^2}\left(\frac{1000\ \text{kg}\cdot\text{m/s}^2}{1\ \text{kN}}\right) = 7.274\ \text{s}^2$$




The time for half of the water in the tank to be discharged ($z = z_0/2$) is

$$\sqrt{\frac{2(3\ \text{m})}{9.81\ \text{m/s}^2} + 7.274\ \text{s}^2} - \sqrt{\frac{2(1.5\ \text{m})}{9.81\ \text{m/s}^2} + 7.274\ \text{s}^2} = \frac{(0.1\ \text{m})^2}{(2\ \text{m})^2}t \quad \rightarrow \quad t = \mathbf{22.0\ s}$$



($b$) Water level after 10s is $\sqrt{\frac{2(3\ \text{m})}{9.81\ \text{m/s}^2} + 7.274\ \text{s}^2} - \sqrt{\frac{2z}{9.81\ \text{m/s}^2} + 7.274\ \text{s}^2} = \frac{(0.1\ \text{m})^2}{(2\ \text{m})^2}(10\ \text{s}) \rightarrow z = \mathbf{2.31\ m}$

***Discussion*** Note that the discharging time is inversely proportional to the square of the orifice diameter. Therefore, the discharging time can be reduced to one-fourth by doubling the diameter of the orifice.

**5-95**

**Solution** Air flows through a pipe that consists of two sections at a specified rate. The differential height of a water manometer placed between the two pipe sections is to be determined.

***Assumptions*** **1**The flow through the pipe is steady, incompressible, and irrotational with negligible friction (so that the Bernoulli equation is applicable). **2** The losses in the reducing section are negligible. **3** The pressure difference across an air column is negligible because of the low density of air, and thus the air column in the manometer can be ignored.

***Properties*** The density of air is given to be $\rho_{air}$ = 1.20 kg/m$^3$. We take the density of water to be $\rho_w$ = 1000 kg/m$^3$.

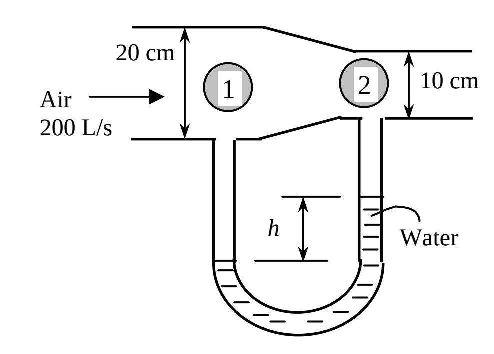
***Analysis*** We take points 1 and 2 along the centerline of the pipe over the two tubes of the manometer. Noting that $z_1 = z_2$ (or, the elevation effects are negligible for gases), the Bernoulli equation between points 1 and 2 gives

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad P_1-P_2=\frac{\rho_{air}(V_2^2-V_1^2)}{2} \qquad (1)$$

We let the differential height of the water manometer be $h$. Then the pressure difference $P_2 - P_1$ can also be expressed as

$$P_1-P_2=\rho_w gh \qquad (2)$$

Combining Eqs. (1) and (2) and solving for $h$,




$$\frac{\rho_{air}(V_2^2-V_1^2)}{2}=\rho_w gh \quad\rightarrow\quad h=\frac{\rho_{air}(V_2^2-V_1^2)}{2g\rho_w}=\frac{V_2^2-V_1^2}{2g\rho_w/\rho_{air}}$$

Calculating the velocities and substituting,

$$V_1=\frac{\dot{V}}{A_1}=\frac{\dot{V}}{\pi D_1^2/4}=\frac{0.2\text{ m}^3/\text{s}}{\pi(0.2\text{ m})^2/4}=6.37\text{ m/s}$$

$$V_2=\frac{\dot{V}}{A_2}=\frac{\dot{V}}{\pi D_2^2/4}=\frac{0.2\text{ m}^3/\text{s}}{\pi(0.1\text{ m})^2/4}=25.5\text{ m/s}$$

$$h=\frac{(25.5\text{ m/s})^2-(6.37\text{ m/s})^2}{2(9.81\text{ m/s}^2)(1000/1.20)}=0.037\text{ m}=\mathbf{3.7\text{ cm}}$$

Therefore, the differential height of the water column will be 3.7 cm.

***Discussion*** Note that the differential height of the manometer is inversely proportional to the density of the manometer fluid. Therefore, heavy fluids such as mercury are used when measuring large pressure differences.

**5-96** **[***Also solved using EES on enclosed DVD***]**

**Solution** Air flows through a horizontal duct of variable cross-section. For a given differential height of a water manometer placed between the two pipe sections, the downstream velocity of air is to be determined, and an error analysis is to be conducted.

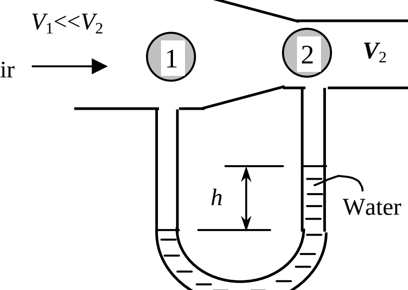
***Assumptions*** **1** The flow through the duct is steady, incompressible, and irrotational with negligible friction (so that the Bernoulli equation is applicable). **2** The losses in this section of the duct are negligible. **3** The pressure difference across an air column is negligible because of the low density of air, and thus the air column in the manometer can be ignored.

***Properties*** The gas constant of air is $R$ = 0.287 kPa·m$^3$/kg·K. We take the density of water to be $\rho_w$ = 1000 kg/m$^3$.

***Analysis*** We take points 1 and 2 along the centerline of the duct over the two tubes of the manometer. Noting that $z_1 = z_2$ (or, the elevation effects are negligible for gases) and $V_1 \cong 0$, the Bernoulli equation between points 1 and 2 gives

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad \frac{P_1}{\rho g}=\frac{P_2}{\rho g}+\frac{V_2^2}{2g} \quad\rightarrow\quad V_2=\sqrt{\frac{2(P_1-P_2)}{\rho_{air}}} \qquad (1)$$

where $P_1 - P_2 = \rho_w gh$

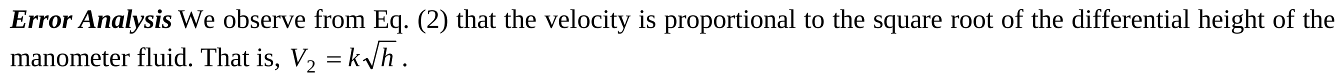



and $$\rho_{air}=\frac{P}{RT}=\frac{100\text{ kPa}}{(0.287\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(298\text{ K})}=1.17\text{ kg/m}^3$$

Substituting into (1), the downstream velocity of air $V_2$ is determined to be

$$V_2=\sqrt{\frac{2\rho_w gh}{\rho_{air}}}=\sqrt{\frac{2(1000\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.08\text{ m})}{1.17\text{ kg/m}^3}}=\mathbf{36.6\text{ m/s}} \qquad (2)$$

Therefore, the velocity of air increases from a low level in the first section to 36.6 m/s in the second section.



***Error Analysis*** We observe from Eq. (2) that the velocity is proportional to the square root of the differential height of the manometer fluid. That is, $V_2 = k\sqrt{h}$ .

Taking the differential: $$dV_2=\tfrac{1}{2}k\frac{dh}{\sqrt{h}}$$

Dividing by $V_2$: $$\frac{dV_2}{V_2}=\tfrac{1}{2}k\frac{dh}{\sqrt{h}}\frac{1}{k\sqrt{h}} \quad\rightarrow\quad \frac{dV_2}{V_2}=\frac{dh}{2h}=\frac{\pm 2\text{ mm}}{2\times 80\text{ mm}}=\mathbf{\pm\ 0.013}$$

Therefore, the uncertainty in the velocity corresponding to an uncertainty of 2 mm in the differential height of water is 1.3%, which corresponds to 0.013×(36.6 m/s) = 0.5m/s. Then the discharge velocity can be expressed as

$$V_2=\mathbf{36.6\pm 0.5\text{ m/s}}$$

***Discussion*** The error analysis does not include the effects of friction in the duct; the error due to frictional losses is most likely more severe than the error calculated here.

**5-97**

**Solution** A tap is opened on the wall of a very large tank that contains air. The maximum flow rate of air through the tap is to be determined, and the effect of a larger diameter lead section is to be assessed.

***Assumptions*** Flow through the tap is steady, incompressible, and irrotational with negligible friction (so that the flow rate is maximum, and the Bernoulli equation is applicable).

***Properties*** The gas constant of air is $R = 0.287$ kPa·m$^3$/kg·K.

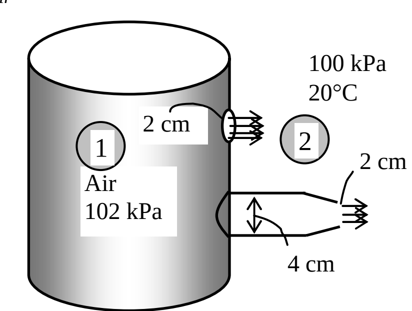
***Analysis*** The density of air in the tank is

$$\rho_{air} = \frac{P}{RT} = \frac{102\ \text{kPa}}{(0.287\ \text{kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(293\ \text{K})} = 1.21\ \text{kg/m}^3$$

We take point 1 in the tank, and point 2 at the exit of the tap along the same horizontal line. Noting that $z_1 = z_2$ (or, the elevation effects are negligible for gases) and $V_1 \cong 0$, the Bernoulli equation between points 1 and 2 gives

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \quad \rightarrow \quad \frac{P_1}{\rho g} = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} \quad \rightarrow \quad V_2 = \sqrt{\frac{2(P_1 - P_2)}{\rho_{air}}}$$

Substituting, the discharge velocity and the flow rate becomes




$$V_2 = \sqrt{\frac{2(P_1 - P_2)}{\rho_{air}}} = \sqrt{\frac{2(102-100)\ \text{kN/m}^2}{1.21\ \text{kg/m}^3}\left(\frac{1000\ \text{kg}\cdot\text{m/s}^2}{1\ \text{kN}}\right)} = 57.5\ \text{m/s}$$

$$\dot{V} = AV_2 = \frac{\pi D_2^2}{4}V_2 = \frac{\pi(0.02\ \text{m})^2}{4}(57.5\ \text{m/s}) = \mathbf{0.0181\ m^3/s}$$

This is the *maximum* flow rate since it is determined by assuming frictionless flow. The actual flow rate will be less.

Adding a 2-m long larger diameter lead section will have **no effect** on the flow rate since the flow is frictionless (by using the Bernoulli equation, it can be shown that the velocity in this section increases, but the pressure decreases, and there is a smaller pressure difference to drive the flow through the tab, with zero net effect on the discharge rate).

***Discussion*** If the pressure in the tank were 300 kPa, the flow is no longer incompressible, and thus the problem in that case should be analyzed using compressible flow theory.

**5-98**

**Solution** Water is flowing through a venturi meter with known diameters and measured pressures. The flow rate of water is to be determined for the case of frictionless flow.

***Assumptions*** **1** The flow through the venturi is steady, incompressible, and irrotational with negligible friction (so that the Bernoulli equation is applicable). **2** The flow is horizontal so that elevation along the centerline is constant. **3** The pressure is uniform at a given cross-section of the venturi meter (or the elevation effects on pressure measurement are negligible).

***Properties*** We take the density of water to be $\rho$ = 1000 kg/m$^3$.

***Analysis*** We take point 1 at the main flow section and point 2 at the throat along the centerline of the venturi meter. Noting that $z_1 = z_2$, the application of the Bernoulli equation between points 1 and 2 gives

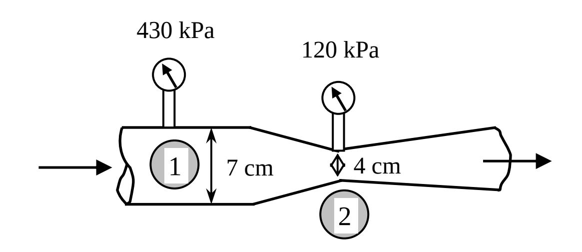
$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad P_1-P_2=\rho\frac{V_2^2-V_1^2}{2} \qquad (1)$$

The flow is assumed to be incompressible and thus the density is constant. Then the conservation of mass relation for this single stream steady flow device can be expressed as

$$\dot{V}_1=\dot{V}_2=\dot{V} \quad\rightarrow\quad A_1V_1=A_2V_2=\dot{V} \quad\rightarrow\quad V_1=\frac{\dot{V}}{A_1} \quad\text{and}\quad V_2=\frac{\dot{V}}{A_2} \qquad (2)$$

Substituting into Eq. (1),

$$P_1-P_2=\rho\frac{(\dot{V}/A_2)^2-(\dot{V}/A_1)^2}{2}=\frac{\rho\dot{V}^2}{2A_2^2}\left(1-\frac{A_2^2}{A_1^2}\right)$$




Solving for $\dot{V}$ gives the desired relation for the flow rate,

$$\dot{V}=A_2\sqrt{\frac{2(P_1-P_2)}{\rho[1-(A_2/A_1)^2]}} \qquad (3)$$

The flow rate for the given case can be determined by substituting the given values into this relation to be

$$\dot{V}=\frac{\pi D_2^2}{4}\sqrt{\frac{2(P_1-P_2)}{\rho[1-(D_2/D_1)^4]}}=\frac{\pi(0.04\text{ m})^2}{4}\sqrt{\frac{2(430-120)\text{ kN/m}^2}{(1000\text{ kg/m}^3)[1\text{-}(4/7)^4]}\left(\frac{1000\text{ kg}\cdot\text{m/s}^2}{1\text{ kN}}\right)}=\mathbf{0.0331\ m^3/s}$$

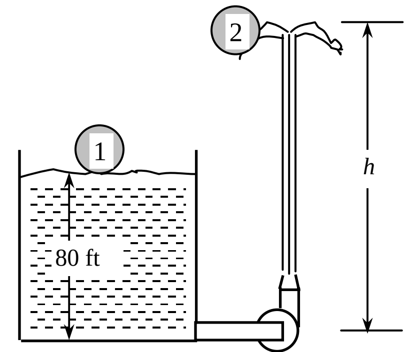
***Discussion*** Venturi meters are commonly used as flow meters to measure the flow rate of gases and liquids by simply measuring the pressure difference $P_1$ **-** $P_2$ by a manometer or pressure transducers. The actual flow rate will be less than the value obtained from Eq. (3) because of the friction losses along the wall surfaces in actual flow. But this difference can be as little as 1% in a well-designed venturi meter. The effects of deviation from the idealized Bernoulli flow can be accounted for by expressing Eq. (3) as

$$\dot{V}=C_dA_2\sqrt{\frac{2(P_1-P_2)}{\rho[1-(A_2/A_1)^2]}}$$

where $C_d$ is the *venturi discharge coefficient* whose value is less than 1 (it is as large as 0.99 for well-designed venturi meters in certain ranges of flow). For Re > $10^5$, the value of venturi discharge coefficient is usually greater than 0.96.

**5-99E**

**Solution** A hose is connected to the bottom of a water tank open to the atmosphere. The hose is equipped with a pump and a nozzle at the end. The maximum height to which the water stream could rise is to be determined.




***Assumptions*** **1** The flow is incompressible with negligible friction. **2** The friction between the water and air is negligible. **3** We take the head loss to be zero ($h_L$ = 0) to determine the maximum rise of water jet.

***Properties*** We take the density of water to be 62.4 lbm/ft$^3$.

***Analysis*** We take point 1 at the free surface of the tank, and point 2 at the top of the water trajectory where $V_2$ = 0. We take the reference level at the bottom of the tank. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad z_1 + h_{\text{pump, u}} = z_2$$

where the useful pump head is

$$h_{\text{pump, u}} = \frac{\Delta P_{\text{pump}}}{\rho g} = \frac{10 \text{ psi}}{(62.4 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)} \left( \frac{144 \text{ lbf/ft}^2}{1 \text{ psi}} \right) \left( \frac{32.2 \text{ lbm} \cdot \text{ft/s}^2}{1 \text{ lbf}} \right) = 23.1 \text{ ft}$$

Substituting, the maximum height rise of water jet from the ground level is determined to be

$$z_2 = z_1 + h_{\text{pump, u}} = 80 + 23.1 = 103.1 \text{ ft} \cong \mathbf{103 \text{ ft}}$$

***Discussion*** The actual rise of water will be less because of the frictional effects between the water and the hose walls and between the water jet and air.

---

**5-100**

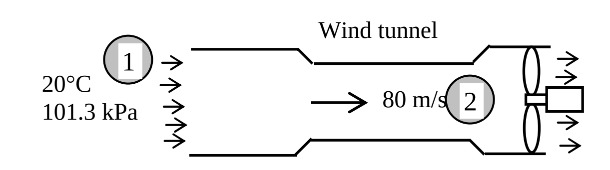
**Solution** A wind tunnel draws atmospheric air by a large fan. For a given air velocity, the pressure in the tunnel is to be determined.

***Assumptions*** **1**The flow through the pipe is steady, incompressible, and irrotational with negligible friction (so that the Bernoulli equation is applicable). **2** Air is and ideal gas.

***Properties*** The gas constant of air is $R$ = 0.287 kPa·m$^3$/kg·K.

***Analysis*** We take point 1 in atmospheric air before it enters the wind tunnel (and thus $P_1 = P_{atm}$ and $V_1 \cong 0$), and point 2 in the wind tunnel. Noting that $z_1 = z_2$ (or, the elevation effects are negligible for gases), the Bernoulli equation between points 1 and 2 gives

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \quad \rightarrow \quad P_2 = P_1 - \frac{\rho V_2^2}{2} \qquad (1)$$




where

$$\rho = \frac{P}{RT} = \frac{101.3 \text{ kPa}}{(0.287 \text{ kPa} \cdot \text{m}^3/\text{kg} \cdot \text{K})(293 \text{ K})} = 1.205 \text{ kg/m}^3$$

Substituting, the pressure in the wind tunnel is determined to be

$$P_2 = (101.3 \text{ kPa}) - (1.205 \text{ kg/m}^3) \frac{(80 \text{ m/s})^2}{2} \left( \frac{1 \text{ N}}{1 \text{ kg} \cdot \text{m/s}^2} \right) \left( \frac{1 \text{ kPa}}{1000 \text{ N/m}^2} \right) = \mathbf{97.4 \text{ kPa}}$$

***Discussion*** Note that the velocity in a wind tunnel increases at the expense of pressure. In reality, the pressure will be even lower because of losses.

---

**5-101**

**Solution** Water flows through the enlargement section of a horizontal pipe at a specified rate. For a given head loss, the pressure change across the enlargement section is to be determined.

***Assumptions*** **1** The flow through the pipe is steady and incompressible. **2** The pipe is horizontal. **3** The kinetic energy correction factors are given to be $\alpha_1 = \alpha_2 = \alpha = 1.05$.

***Properties*** We take the density of water to be $\rho = 1000$ kg/m$^3$.

***Analysis*** We take points 1 and 2 at the inlet and exit of the enlargement section along the centerline of the pipe. Noting that $z_1 = z_2$, the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad P_2 - P_1 = \rho \frac{\alpha(V_1^2 - V_2^2)}{2} - \rho g h_L$$

where the inlet and exit velocities are

$$V_1 = \frac{\dot{V}}{A_1} = \frac{\dot{V}}{\pi D_1^2/4} = \frac{0.025 \text{ m}^3/\text{s}}{\pi (0.06 \text{ m})^2/4} = 8.84 \text{ m/s}$$

$$V_2 = \frac{\dot{V}}{A_2} = \frac{\dot{V}}{\pi D_2^2/4} = \frac{0.025 \text{ m}^3/\text{s}}{\pi (0.11 \text{ m})^2/4} = 2.63 \text{ m/s}$$

Water
0.025 m$^3$/s
1
6 cm
2
11 cm

Substituting, the change in static pressure across the enlargement section is determined to be

$$P_2 - P_1 = (1000 \text{ kg/m}^3)\left(\frac{1.05[(8.84 \text{ m/s})^2 - (2.63 \text{ m/s})^2]}{2} - (9.81 \text{ m/s}^2)(0.45 \text{ m})\right)\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ kPa}}{1000 \text{ N/m}^2}\right)$$

$$= \mathbf{33.0 \text{ kPa}}$$

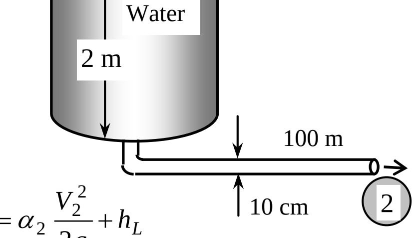
Therefore, the water pressure increases by 33 kPa across the enlargement section.

***Discussion*** Note that the pressure increases despite the head loss in the enlargement section. This is due to dynamic pressure being converted to static pressure. But the total pressure (static + dynamic) decreases by 0.45 m (or 4.41 kPa) as a result of frictional effects.

---

**5-102**

**Solution** A water tank open to the atmosphere is initially filled with water. A sharp-edged orifice at the bottom drains to the atmosphere through a long pipe with a specified head loss. The initial discharge velocity is to be determined.




***Assumptions*** **1** The flow is incompressible. **2** The draining pipe is horizontal. **3** There are no pumps or turbines in the system. **4** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Analysis*** We take point 1 at the free surface of the tank, and point 2 at the exit of the pipe. We take the reference level at the centerline of the orifice ($z_2 =$ 0. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the energy equation between these two points (in terms of heads) simplifies to

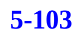
$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad z_1 = \alpha_2 \frac{V_2^2}{2g} + h_L$$

where $\alpha_2 = 1$ and the head loss is given to be $h_L = 1.5$ m. Solving for $V_2$ and substituting, the discharge velocity of water is determined to be

$$V_2 = \sqrt{2g(z_1 - h_L)} = \sqrt{2(9.81 \text{ m/s}^2)(2 - 1.5) \text{ m}} = \mathbf{3.13 \text{ m/s}}$$

***Discussion*** Note that this is the discharge velocity at the beginning, and the velocity will decrease as the water level in the tank drops. The head loss in that case will change since it depends on velocity.

---



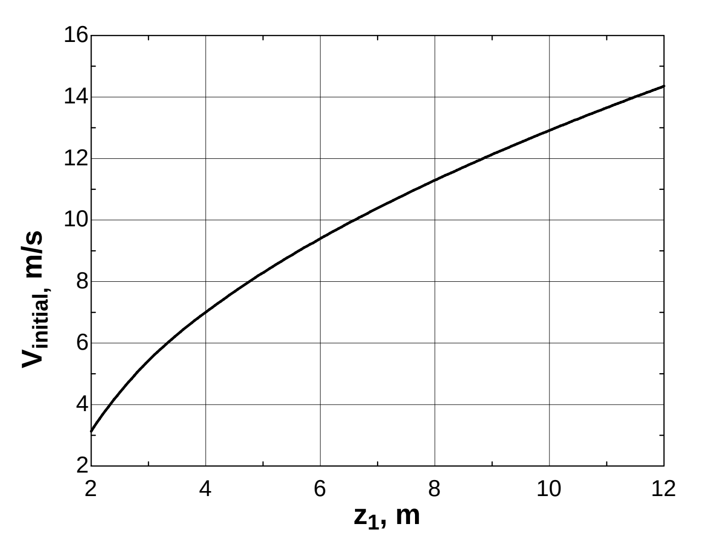
**5-103** 

**Solution** The previous problem is reconsidered. The effect of the tank height on the initial discharge velocity of water from the completely filled tank as the tank height varies from 2 to 20 m in increments of 2 m at constant heat loss is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
g=9.81 "m/s2"
rho=1000 "kg/m3"
h_L=1.5 "m"
D=0.10 "m"
V_initial=SQRT(2*g*(z1-h_L)) "m/s"
```

| Tank height, $z1$, m | Head Loss, $h_L$, m | Initial velocity $V_{initial}$, m/s |
|---|---|---|
| 2 | 1.5 | 3.13 |
| 3 | 1.5 | 5.42 |
| 4 | 1.5 | 7.00 |
| 5 | 1.5 | 8.29 |
| 6 | 1.5 | 9.40 |
| 7 | 1.5 | 10.39 |
| 8 | 1.5 | 11.29 |
| 9 | 1.5 | 12.13 |
| 10 | 1.5 | 12.91 |
| 11 | 1.5 | 13.65 |
| 12 | 1.5 | 14.35 |




***Discussion*** The dependence of $V$ on height is not linear, but rather $V$ changes as the square root of $z_1$.

**5-104**

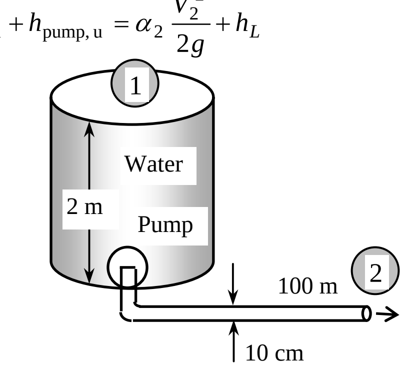
**Solution** A water tank open to the atmosphere is initially filled with water. A sharp-edged orifice at the bottom drains to the atmosphere through a long pipe equipped with a pump with a specified head loss. The required pump head to assure a certain velocity is to be determined.

***Assumptions*** **1** The flow is incompressible. **2** The draining pipe is horizontal. **3** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Analysis*** We take point 1 at the free surface of the tank, and point 2 at the exit of the pipe. We take the reference level at the centerline of the orifice ($z_2 = 0$), and take the positive direction of $z$ to be upwards. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low $(V_1 \cong 0)$, the energy equation between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad z_1+h_{\text{pump, u}}=\alpha_2\frac{V_2^2}{2g}+h_L$$

where $\alpha_2 = 1$ and the head loss is given to be $h_L = 1.5$ m. Solving for $h_{\text{pump, u}}$ and substituting, the required useful pump head is determined to be




$$h_{\text{pump, u}}=\sqrt{\frac{V_2^2}{2g}-z_1+h_L}=\sqrt{\frac{(6\text{ m/s})^2}{2(9.81\text{ m/s}^2)}-(2\text{ m})+(1.5\text{ m})}=\mathbf{1.15\ m}$$

***Discussion*** Note that this is the required useful pump head at the beginning, and it will need to be increased as the water level in the tank drops to make up for the lost elevation head to maintain the constant discharge velocity.

---

## Design and Essay Problems

**5-105 to 5-109**

**Solution** Students' essays and designs should be unique and will differ from each other.

---

**5-110**

**Solution** We are to evaluate a proposed modification to a wind turbine.

***Analysis*** Using the mass and the Bernoulli equations, it can be shown that **this is a bad idea** – the velocity at the exit of nozzle is equal to the wind velocity. (The velocity at nozzle inlet is much lower). Sample calculation using EES using a wind velocity of 10 m/s:

```
V0=10 "m/s"
rho=1.2 "kg/m3"
g=9.81 "m/s2"
A1=2 "m2"
A2=1 "m2"
A1*V1=A2*V2
P1/rho+V1^2/2=V2^2/2
m=rho*A1*V1
m*V0^2/2=m*V2^2/2
```

$V_0$ =10 m/s

$V_1$ =5 m/s

$V_2$ =10 m/s

***Results***: $V_1$ = 5 m/s, $V_2$ = 10 m/s, $m$ = 12 kg/s (mass flow rate).

***Discussion*** Students' approaches may be different, but they should come to the same conclusion – this does not help.



# Solutions Manual for

# Fluid Mechanics: Fundamentals and Applications

# by Çengel & Cimbala

---

# CHAPTER 6
# MOMENTUM ANALYSIS OF FLOW SYSTEMS

---

## Newton's Laws and Conservation of Momentum

### 6-1C

**Solution** We are to express Newton's three laws.

***Analysis*** *Newton's first law* states that "**a body at rest remains at rest, and a body in motion remains in motion at the same velocity in a straight path when the net force acting on it is zero.**" Therefore, a body tends to preserve its state or inertia. *Newton's second law* states that "**the acceleration of a body is proportional to the net force acting on it and is inversely proportional to its mass.**" *Newton's third law states "**when a body exerts a force on a second body, the second body exerts an equal and opposite force on the first.**"*

***Discussion*** As we shall see in later chapters, the differential equation of fluid motion is based on Newton's second law.

### 6-2C

**Solution** We are to discuss if momentum is a vector, and its direction.

***Analysis*** Since momentum ( $m\vec{V}$ ) is the product of a vector (velocity) and a scalar (mass), **momentum must be a vector that points in the same direction as the velocity vector**.

***Discussion*** In the general case, we must solve three components of the linear momentum equation, since it is a vector equation.

### 6-3C

**Solution** We are to discuss the conservation of momentum principle.

***Analysis*** The *conservation of momentum principle* is expressed as "**the momentum of a system remains constant when the net force acting on it is zero, and thus the momentum of such systems is conserved**". The momentum of a body remains constant if the net force acting on it is zero.

***Discussion*** Momentum is not conserved in general, because when we apply a force, the momentum changes.

### 6-4C

**Solution** We are to discuss Newton's second law for rotating bodies.

***Analysis*** Newton's second law of motion, also called the *angular momentum equation*, is expressed as "**the rate of change of the angular momentum of a body is equal to the net torque acting on it**." For a non-rigid body with zero net torque, the angular momentum remains constant, but the **angular velocity changes in accordance with** ***$I\omega$ = constant*** where $I$ is the moment of inertia of the body.

***Discussion*** Angular momentum is analogous to linear momentum in this way: Linear momentum does not change unless a force acts on it. Angular momentum does not change unless a torque acts on it.

### 6-5C

**Solution** We are to compare the angular momentum of two rotating bodies

***Analysis*** **No. The two bodies do not necessarily have the same angular momentum**. Two rigid bodies having the same mass and angular speed may have different angular momentums unless they also have the same moment of inertia $I$.

***Discussion*** The reason why flywheels have most of their mass at the outermost radius, is to maximize the angular momentum.

## Linear Momentum Equation

### 6-6C

**Solution** We are to discuss the importance of the RTT, and its relationship to the linear momentum equation.

***Analysis*** **The relationship between the time rates of change of an extensive property for a system and for a control volume** is expressed by the *Reynolds transport theorem* (RTT), which provides the link between the system and control volume concepts. The linear momentum equation is obtained by **setting** $b = \vec{V}$ **and thus** $B = m\vec{V}$ **in the Reynolds transport theorem**.

***Discussion*** Newton's second law applies directly to a system of fixed mass, but we use the RTT to transform from the system formulation to the control volume formulation.

### 6-7C

**Solution** We are to describe and discuss body forces and surface forces.

***Analysis*** The forces acting on the control volume consist of *body forces* that **act throughout the entire body of the control volume** (such as gravity, electric, and magnetic forces) and *surface forces* that **act on the control surface** (such as the pressure forces and reaction forces at points of contact). The *net force* acting on a control volume is the **sum of all body *and* surface forces**. Fluid **weight is a body force**, and **pressure is a surface force** (acting per unit area).

***Discussion*** In a general fluid flow, the flow is influenced by both body and surface forces.

### 6-8C

**Solution** We are to discuss surface forces in a control volume analysis.

***Analysis*** All surface forces arise as **the control volume is isolated from its surroundings for analysis, and the effect of any detached object is accounted for by a force at that location**. We can minimize the number of surface forces exposed by **choosing the control volume (wisely) such that the forces that we are not interested in remain internal, and thus they do not complicate the analysis**. A well-chosen control volume exposes only the forces that are to be determined (such as reaction forces) and a minimum number of other forces.

***Discussion*** There are many choices of control volume for a given problem. Although there are not really "wrong" and "right" choices of control volume, there certainly are "wise" and "unwise" choices of control volume.

### 6-9C

**Solution** We are to discuss the momentum flux correction factor, and its significance.

***Analysis*** The *momentum-flux correction factor* $\beta$ enables us to express the momentum flux in terms of the mass flow rate and mean flow velocity as $\boxed{\int_{A_c} \rho\vec{V}(\vec{V}\cdot\vec{n})dA_c = \beta\dot{m}\vec{V}_{avg}}$. The value of $\beta$ is unity for uniform flow, such as a jet flow, nearly unity for fully developed turbulent pipe flow (between 1.01 and 1.04), but about 1.3 for fully developed laminar pipe flow. **So it is significant and should be considered in laminar flow**; it is often ignored in turbulent flow.

***Discussion*** Even though $\beta$ is nearly unity for many turbulent flows, it is wise not to ignore it.

**6-10C**

**Solution** We are to discuss the momentum equation for steady one-D flow with no external forces.

***Analysis*** The momentum equation for steady one-dimensional flow for the case of no external forces is

$$\sum \vec{F} = 0 = \sum_{\text{out}} \beta \dot{m} \vec{V} - \sum_{\text{in}} \beta \dot{m} \vec{V}$$

where the left hand side is the **net force acting on the control volume** (which is zero here), the first term on the right hand side is the **incoming momentum flux**, and the second term on the right is the **outgoing momentum flux** by mass.

***Discussion*** This is a special simplified case of the more general momentum equation, since there are no forces. In this case we can say that momentum is conserved.

---

**6-11C**

**Solution** We are to explain why we can usually work with gage pressure rather than absolute pressure.

***Analysis*** In the application of the momentum equation, we can disregard the atmospheric pressure and work with gage pressures only since the **atmospheric pressure acts in all directions**, and its effect cancels out in every direction.

***Discussion*** In some applications, it is better to use absolute pressure everywhere, but for most practical engineering problems, it is simpler to use gage pressure everywhere, with no loss of accuracy.

---

**6-12C**

**Solution** We are to compare the reaction force on two fire hoses.

***Analysis*** **The fireman who holds the hose backwards so that the water makes a U-turn before being discharged will experience a greater reaction force**. This is because of the vector nature of the momentum equation. Specifically, the inflow and outflow terms end up with the same sign (they add together) for the case with the U-turn, whereas they have opposite signs (one partially cancels out the other) for the case without the U-turn.

***Discussion*** Direction is not an issue with the conservation of mass or energy equations, since they are scalar equations.

---

**6-13C**

**Solution** We are to discuss if the upper limit of a rocket's velocity is limited to *V*, its discharge velocity.

***Analysis*** **No, *V* is not the upper limit to the rocket's ultimate velocity**. Without friction the rocket velocity will continue to increase (i.e., it will continue to accelerate) as more gas is expelled out the nozzle.

***Discussion*** This is a simple application of Newton's second law. As long as there is a force acting on the rocket, it will continue to accelerate, regardless of how that force is generated.

---

**6-14C**

**Solution** We are to describe how a helicopter can hover.

***Analysis*** A helicopter hovers because **the strong downdraft of air, caused by the overhead propeller blades, manifests a momentum in the air stream**. This momentum must be countered by the helicopter lift force.

***Discussion*** In essence, the helicopter stays aloft by pushing down on the air with a net force equal to its weight.

---

**6-15C**

**Solution** We are to discuss the power required for a helicopter to hover at various altitudes.

***Analysis*** Since the air density decreases, **it requires more energy for a helicopter to hover at higher altitudes**, because more air must be forced into the downdraft by the helicopter blades to provide the same lift force. Therefore, it takes more power for a helicopter to hover on the top of a high mountain than it does at sea level.

***Discussion*** This is consistent with the limiting case – if there were no air, the helicopter would not be able to hover at all. There would be no air to push down.

---

**6-16C**

**Solution** We are to discuss helicopter performance in summer versus winter.

***Analysis*** In winter the air is generally colder, and thus denser. Therefore, less air must be driven by the blades to provide the same helicopter lift, requiring less power. **Less energy is required in the winter**.

***Discussion*** However, it is also harder for the blades to move through the denser cold air, so there is more torque required of the engine in cold weather.

---

**6-17C**

**Solution** We are to discuss if the force required to hold a plate stationary doubles when the jet velocity doubles.

***Analysis*** **No, the force will not double**. In fact, the force required to hold the plate against the horizontal water stream **will increase by a factor of 4** when the velocity is doubled since

$$F = \dot{m}V = (\rho AV)V = \rho AV^2$$

and thus the *force is proportional to the square of the velocity*.

***Discussion*** You can think of it this way: Since momentum flux is mass flow rate times velocity, a doubling of the velocity doubles both the mass flow rate *and* the velocity, increasing the momentum flux by a factor of four.

---

**6-18C**

**Solution** We are to discuss the acceleration of a cart hit by a water jet.

***Analysis*** **The acceleration is not be constant since the force is not constant**. The impulse force exerted by the water on the plate is $F = \dot{m}V = (\rho AV)V = \rho AV^2$, where $V$ is the relative velocity between the water and the plate, which is moving. The magnitude of the plate acceleration is thus $a = F/m$**.** But as the plate begins to move, $V$ decreases, so the acceleration must also decrease.

***Discussion*** It is the *relative* velocity of the water jet on the cart that contributes to the cart's acceleration.

---

**6-19C**

**Solution** We are to discuss the maximum possible velocity of a cart hit by a water jet.

***Analysis*** The **maximum possible velocity for the plate is the velocity of the water jet**. As long as the plate is moving slower than the jet, the water exerts a force on the plate, which causes it to accelerate, until terminal jet velocity is reached.

***Discussion*** Once the relative velocity is zero, the jet supplies no force to the cart, and thus it cannot accelerate further.

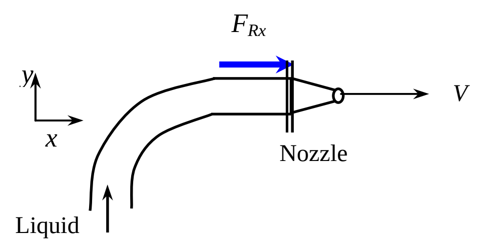
---

## 6-20

**Solution** It is to be shown that the force exerted by a liquid jet of velocity $V$ on a stationary nozzle is proportional to $V^2$, or alternatively, to $\dot{m}^2$.

***Assumptions*** **1** The flow is steady and incompressible. **2** The nozzle is given to be stationary. **3** The nozzle involves a 90° turn and thus the incoming and outgoing flow streams are normal to each other. **4** The water is discharged to the atmosphere, and thus the gage pressure at the outlet is zero.

***Analysis*** We take the nozzle as the control volume, and the flow direction at the outlet as the $x$ axis. Note that the nozzle makes a 90° turn, and thus it does not contribute to any pressure force or momentum flux term at the inlet in the $x$ direction. Noting that $\dot{m} = \rho AV$ where $A$ is the nozzle outlet area and $V$ is the average nozzle outlet velocity, the momentum equation for steady one-dimensional flow in the $x$ direction reduces to




$$\sum \vec{F} = \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V} \quad \rightarrow \quad F_{Rx} = \beta \dot{m}_{out} V_{out} = \beta \dot{m} V$$

where $F_{Rx}$ is the reaction force on the nozzle due to liquid jet at the nozzle outlet. Then,

$$\dot{m} = \rho AV \quad \rightarrow \quad F_{Rx} = \beta \dot{m} V = \beta \rho AVV = \beta \rho AV^2 \quad \text{or} \quad \boxed{F_{Rx} = \beta \dot{m} V = \beta \dot{m}\frac{\dot{m}}{\rho A} = \beta \frac{\dot{m}^2}{\rho A}}$$

Therefore, **the force exerted by a liquid jet of velocity *V* on this stationary nozzle is proportional to $V^2$, or alternatively, to $\dot{m}^2$.**

***Discussion*** If there were not a 90° turn, we would need to take into account the momentum flux and pressure contributions at the inlet.

---

## 6-21

**Solution** A water jet of velocity $V$ impinges on a plate moving toward the water jet with velocity ½$V$. The force required to move the plate towards the jet is to be determined in terms of $F$ acting on the stationary plate.

***Assumptions*** **1** The flow is steady and incompressible. **2** The plate is vertical and the jet is normal to plate. **3** The pressure on both sides of the plate is atmospheric pressure (and thus its effect cancels out). **4** Fiction during motion is negligible. **5** There is no acceleration of the plate. **6** The water splashes off the sides of the plate in a plane normal to the jet. **6** Jet flow is nearly uniform and thus the effect of the momentum-flux correction factor is negligible, $\beta \cong 1$.

***Analysis*** We take the plate as the control volume. The relative velocity between the plate and the jet is $V$ when the plate is stationary, and 1.5$V$ when the plate is moving with a velocity ½$V$ towards the plate. Then the momentum equation for steady one-dimensional flow in the horizontal direction reduces to

1/2V
V
Waterjet

$$\sum \vec{F} = \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V} \quad \rightarrow \quad -F_R = -\dot{m}_i V_i \quad \rightarrow \quad F_R = \dot{m}_i V_i$$

*Stationary plate*: $(V_i = V \;\text{ and }\; \dot{m}_i = \rho AV_i = \rho AV) \rightarrow F_R = \rho AV^2 = F$

*Moving plate*: $(V_i = 1.5V \;\text{ and }\; \dot{m}_i = \rho AV_i = \rho A(1.5V))$

$$\rightarrow F_R = \rho A(1.5V)^2 = 2.25\rho AV^2 = 2.25F$$

Therefore, the force required to hold the plate stationary against the oncoming water jet becomes **2.25 times greater** when the jet velocity becomes 1.5 times greater.

***Discussion*** Note that when the plate is stationary, $V$ is also the jet velocity. But if the plate moves toward the stream with velocity ½$V$, then the *relative* velocity is 1.5$V$, and the amount of mass striking the plate (and falling off its sides) per unit time also increases by 50%.

---

**6-22**

**Solution** A 90° elbow deflects water upwards and discharges it to the atmosphere at a specified rate. The gage pressure at the inlet of the elbow and the anchoring force needed to hold the elbow in place are to be determined.

***Assumptions*** **1** The flow is steady, frictionless, incompressible, and irrotational (so that the Bernoulli equation is applicable). **2** The weight of the elbow and the water in it is negligible. **3** The water is discharged to the atmosphere, and thus the gage pressure at the outlet is zero. **4** The momentum-flux correction factor for each inlet and outlet is given to be $\beta = 1.03$.

***Properties*** We take the density of water to be 1000 kg/m³.

***Analysis*** (*a*) We take the elbow as the control volume, and designate the entrance by 1 and the outlet by 2. We also designate the horizontal coordinate by *x* (with the direction of flow as being the positive direction) and the vertical coordinate by *z*. The continuity equation for this one-inlet one-outlet steady flow system is $\dot{m}_1 = \dot{m}_2 = \dot{m} = 30 \text{ kg/s}$. Noting that $\dot{m} = \rho AV$, the mean inlet and outlet velocities of water are

$$V_1 = V_2 = V = \frac{\dot{m}}{\rho A} = \frac{\dot{m}}{\rho(\pi D^2/4)} = \frac{25 \text{ kg/s}}{(1000 \text{ kg/m}^3)[\pi(0.1\text{ m})^2/4]} = 3.18 \text{ m/s}$$

Noting that $V_1 = V_2$ and $P_2 = P_{atm}$, the Bernoulli equation for a streamline going through the center of the reducing elbow is expressed as

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \;\rightarrow\; P_1 - P_2 = \rho g(z_2 - z_1) \;\rightarrow\; P_{1,\text{gage}} = \rho g(z_2 - z_1)$$

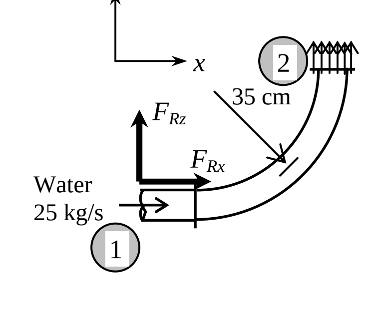
Substituting,

$$P_{1,\text{ gage}} = (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(0.35 \text{ m})\left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right) = 3.434 \text{ kN/m}^2 = 3.434 \text{ kPa} \cong \mathbf{3.43\ kPa}$$

(*b*) The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta\dot{m}\vec{V} - \sum_{\text{in}} \beta\dot{m}\vec{V}$. We let the *x*- and *z*-components of the anchoring force of the elbow be $F_{Rx}$ and $F_{Rz}$, and assume them to be in the positive directions. We also use gage pressures to avoid dealing with the atmospheric pressure which acts on all surfaces. Then the momentum equations along the *x* and *y* axes become

$$F_{Rx} + P_{1,\text{gage}}A_1 = 0 - \beta\dot{m}(+V_1) = -\beta\dot{m}V$$

$$F_{Rz} = \beta\dot{m}(+V_2) = \beta\dot{m}V$$




Solving for $F_{Rx}$ and $F_{Rz}$, and substituting the given values,

$$F_{Rx} = -\beta\dot{m}V - P_{1,\text{ gage}}A_1$$

$$= -1.03(25 \text{ kg/s})(3.18 \text{ m/s})\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) - (3434 \text{ N/m}^2)[\pi(0.1\text{ m})^2/4]$$

$$= -109 \text{ N}$$

$$F_{Ry} = \beta\dot{m}V = 1.03(25 \text{ kg/s})(3.18 \text{ m/s})\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 81.9 \text{ N}$$

and $F_R = \sqrt{F_{Rx}^2 + F_{Ry}^2} = \sqrt{(-109)^2 + 81.9^2} = \mathbf{136\ N}$, $\theta = \tan^{-1}\frac{F_{Ry}}{F_{Rx}} = \tan^{-1}\frac{81.9}{-109} = -37° = \mathbf{143°}$

***Discussion*** Note that the magnitude of the anchoring force is 136 N, and its line of action makes 143° from the positive *x* direction. Also, a negative value for $F_{Rx}$ indicates the assumed direction is wrong, and should be reversed.

**6-23**

**Solution** A 180° elbow forces the flow to make a U-turn and discharges it to the atmosphere at a specified rate. The gage pressure at the inlet of the elbow and the anchoring force needed to hold the elbow in place are to be determined.

***Assumptions*** **1** The flow is steady, frictionless, one-dimensional, incompressible, and irrotational (so that the Bernoulli equation is applicable). **2** The weight of the elbow and the water in it is negligible. **3** The water is discharged to the atmosphere, and thus the gage pressure at the outlet is zero. **4** The momentum-flux correction factor for each inlet and outlet is given to be $\beta = 1.03$.

***Properties*** We take the density of water to be 1000 kg/m³.

***Analysis*** (*a*) We take the elbow as the control volume, and designate the entrance by 1 and the outlet by 2. We also designate the horizontal coordinate by *x* (with the direction of flow as being the positive direction) and the vertical coordinate by *z*. The continuity equation for this one-inlet one-outlet steady flow system is $\dot{m}_1 = \dot{m}_2 = \dot{m} = 30\text{ kg/s}$. Noting that $\dot{m} = \rho AV$, the mean inlet and outlet velocities of water are

$$V_1 = V_2 = V = \frac{\dot{m}}{\rho A} = \frac{\dot{m}}{\rho(\pi D^2/4)} = \frac{25\text{ kg/s}}{(1000\ \text{kg/m}^3)[\pi(0.1\text{ m})^2/4]} = 3.18\text{ m/s}$$

Noting that $V_1 = V_2$ and $P_2 = P_{atm}$, the Bernoulli equation for a streamline going through the center of the reducing elbow is expressed as

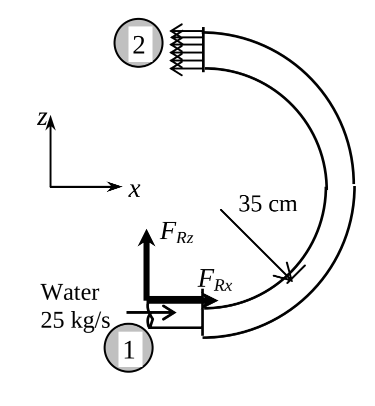
$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \ \rightarrow\ P_1 - P_2 = \rho g(z_2 - z_1) \ \rightarrow\ P_{1,\text{ gage}} = \rho g(z_2 - z_1)$$

Substituting,

$$P_{1,\text{ gage}} = (1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(0.70\ \text{m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 6.867\ \text{kN/m}^2 = 6.867\ \text{kPa} \cong \mathbf{6.87\ kPa}$$

(*b*) The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta\dot{m}\vec{V} - \sum_{\text{in}} \beta\dot{m}\vec{V}$. We let the *x*- and *z*-components of the anchoring force of the elbow be $F_{Rx}$ and $F_{Rz}$, and assume them to be in the positive directions. We also use gage pressures to avoid dealing with the atmospheric pressure which acts on all surfaces. Then the momentum equations along the *x* and *z* axes become

$$F_{Rx} + P_{1,\text{gage}}A_1 = \beta\dot{m}(-V_2) - \beta\dot{m}(+V_1) = -2\beta\dot{m}V$$
$$F_{Rz} = 0$$




Solving for $F_{Rx}$ and substituting the given values,

$$F_{Rx} = -2\beta\dot{m}V - P_{1,\text{ gage}}A_1$$
$$= -2\times 1.03(25\text{ kg/s})(3.18\text{ m/s})\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) - (6867\text{ N/m}^2)[\pi(0.1\text{ m})^2/4]$$
$$= -218\text{ N}$$

and $F_R = F_{Rx}$ **= - 218 N** since the *y*-component of the anchoring force is zero. Therefore, the anchoring force has a magnitude of 218 N and it acts in the negative *x* direction.

***Discussion*** Note that a negative value for $F_{Rx}$ indicates the assumed direction is wrong, and should be reversed.

**6-24E**

**Solution** A horizontal water jet strikes a vertical stationary plate normally at a specified velocity. For a given anchoring force needed to hold the plate in place, the flow rate of water is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The water splatters off the sides of the plate in a plane normal to the jet. **3** The water jet is exposed to the atmosphere, and thus the pressure of the water jet and the splattered water is the atmospheric pressure which is disregarded since it acts on the entire control surface. **4** The vertical forces and momentum fluxes are not considered since they have no effect on the horizontal reaction force. **5** Jet flow is nearly uniform and thus the effect of the momentum-flux correction factor is negligible, $\beta \cong 1$.

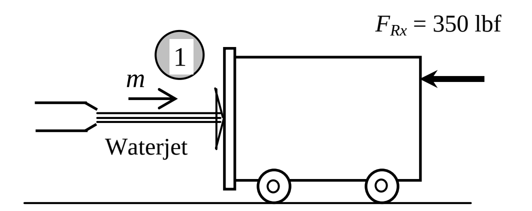
***Properties*** We take the density of water to be 62.4 lbm/ft$^3$.

***Analysis*** We take the plate as the control volume such that it contains the entire plate and cuts through the water jet and the support bar normally, and the direction of flow as the positive direction of $x$ axis. The momentum equation for steady one-dimensional flow in the $x$ (flow) direction reduces in this case to

$$\sum \vec{F} = \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V} \quad \rightarrow \quad -F_{Rx} = -\dot{m}V_1 \quad \rightarrow \quad F_R = \dot{m}V_1$$

We note that the reaction force acts in the opposite direction to flow, and we should not forget the negative sign for forces and velocities in the negative $x$-direction. Solving for $\dot{m}$ and substituting the given values,

$$\dot{m} = \frac{F_{Rx}}{V_1} = \frac{350\text{ lbf}}{30\text{ ft/s}}\left(\frac{32.2\text{ lbm}\cdot\text{ft/s}^2}{1\text{ lbf}}\right) = 376\text{ lbm/s}$$




Then the volume flow rate becomes

$$\dot{V} = \frac{\dot{m}}{\rho} = \frac{376\text{ lbm/s}}{62.4\text{ lbm/ft}^3} = \mathbf{6.02\ ft^3/s}$$

Therefore, the volume flow rate of water under stated assumptions must be 6.02 ft$^3$/s.

***Discussion*** In reality, some water will be scattered back, and this will add to the reaction force of water. The flow rate in that case will be less.

**6-25**

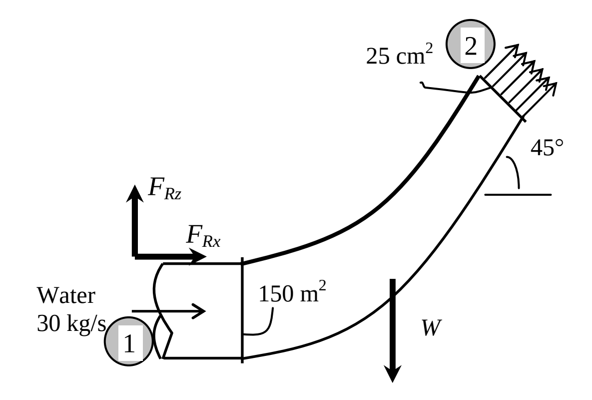
**Solution** A reducing elbow deflects water upwards and discharges it to the atmosphere at a specified rate. The anchoring force needed to hold the elbow in place is to be determined.

***Assumptions*** **1** The flow is steady, frictionless, one-dimensional, incompressible, and irrotational (so that the Bernoulli equation is applicable). **2** The weight of the elbow and the water in it is considered. **3** The water is discharged to the atmosphere, and thus the gage pressure at the outlet is zero. **4** The momentum-flux correction factor for each inlet and outlet is given to be $\beta = 1.03$.

***Properties*** We take the density of water to be 1000 kg/m³.

***Analysis*** The weight of the elbow and the water in it is

$$W = mg = (50\ \text{kg})(9.81\ \text{m/s}^2) = 490.5\ \text{N} = 0.4905\ \text{kN}$$




We take the elbow as the control volume, and designate the entrance by 1 and the outlet by 2. We also designate the horizontal coordinate by $x$ (with the direction of flow as being the positive direction) and the vertical coordinate by $z$. The continuity equation for this one-inlet one-outlet steady flow system is $\dot m_1 = \dot m_2 = \dot m = 30\ \text{kg/s}$. Noting that $\dot m = \rho AV$, the inlet and outlet velocities of water are

$$V_1 = \frac{\dot m}{\rho A_1} = \frac{30\ \text{kg/s}}{(1000\ \text{kg/m}^3)(0.0150\ \text{m}^2)} = 2.0\ \text{m/s}$$

$$V_2 = \frac{\dot m}{\rho A_2} = \frac{30\ \text{kg/s}}{(1000\ \text{kg/m}^3)(0.0025\ \text{m}^2)} = 12\ \text{m/s}$$

Taking the center of the inlet cross section as the reference level ($z_1 = 0$) and noting that $P_2 = P_{atm}$, the Bernoulli equation for a streamline going through the center of the reducing elbow is expressed as

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \ \rightarrow\ P_1 - P_2 = \rho g\left(\frac{V_2^2 - V_1^2}{2g} + z_2 - z_1\right) \ \rightarrow\ P_{1,\text{ gage}} = \rho g\left(\frac{V_2^2 - V_1^2}{2g} + z_2\right)$$

Substituting,

$$P_{1,\text{ gage}} = (1000\ \text{kg/m}^3)(9.81\ \text{m/s}^2)\left(\frac{(12\ \text{m/s})^2 - (2\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)} + 0.4\right)\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) = 73.9\ \text{kN/m}^2 = 73.9\ \text{kPa}$$

The momentum equation for steady one-dimensional flow is $\sum \vec F = \sum_{\text{out}} \beta \dot m \vec V - \sum_{\text{in}} \beta \dot m \vec V$. We let the $x$- and $z$- components of the anchoring force of the elbow be $F_{Rx}$ and $F_{Rz}$, and assume them to be in the positive directions. We also use gage pressures to avoid dealing with the atmospheric pressure which acts on all surfaces. Then the momentum equations along the $x$ and $z$ axes become

$$F_{Rx} + P_{1,\text{gage}} A_1 = \beta \dot m V_2 \cos\theta - \beta \dot m V_1 \quad \text{and} \quad F_{Rz} - W = \beta \dot m V_2 \sin\theta$$

Solving for $F_{Rx}$ and $F_{Rz}$, and substituting the given values,

$$F_{Rx} = \beta \dot m (V_2 \cos\theta - V)_1 - P_{1,\text{ gage}} A_1 = 1.03(30\ \text{kg/s})[(12\cos 45°\text{-}2)\ \text{m/s}]\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) - (73.9\ \text{kN/m}^2)(0.0150\ \text{m}^2)$$

$$= -0.908\ \text{kN}$$

$$F_{Rz} = \beta \dot m V_2 \sin\theta + W = 1.03(30\ \text{kg/s})(12\sin 45°\ \text{m/s})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) + 0.4905\ \text{kN} = 0.753\ \text{kN}$$

$$F_R = \sqrt{F_{Rx}^2 + F_{Rz}^2} = \sqrt{(-0.908)^2 + (0.753)^2} = \mathbf{1.18\ kN}, \quad \theta = \tan^{-1}\frac{F_{Rz}}{F_{Rx}} = \tan^{-1}\frac{0.753}{-0.908} = \mathbf{-39.7°}$$

***Discussion*** Note that the magnitude of the anchoring force is 1.18 kN, and its line of action makes –39.7° from +$x$ direction. Negative value for $F_{Rx}$ indicates the assumed direction is wrong.

---

**6-26**

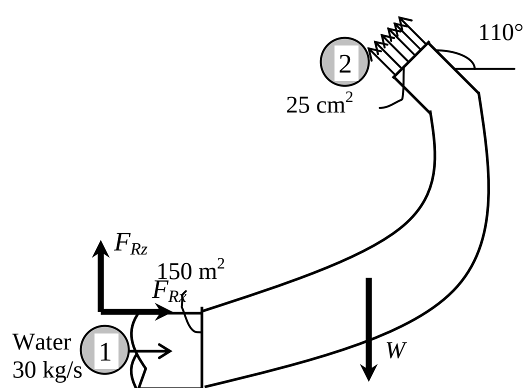
**Solution** A reducing elbow deflects water upwards and discharges it to the atmosphere at a specified rate. The anchoring force needed to hold the elbow in place is to be determined.

***Assumptions*** **1** The flow is steady, frictionless, one-dimensional, incompressible, and irrotational (so that the Bernoulli equation is applicable). **2** The weight of the elbow and the water in it is considered. **3** The water is discharged to the atmosphere, and thus the gage pressure at the outlet is zero. **4** The momentum-flux correction factor for each inlet and outlet is given to be $\beta = 1.03$.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** The weight of the elbow and the water in it is

$$W = mg = (50\text{ kg})(9.81\text{ m/s}^2) = 490.5\text{ N} = 0.4905\text{ kN}$$




We take the elbow as the control volume, and designate the entrance by 1 and the outlet by 2. We also designate the horizontal coordinate by $x$ (with the direction of flow as being the positive direction) and the vertical coordinate by $z$. The continuity equation for this one-inlet one-outlet steady flow system is $\dot m_1 = \dot m_2 = \dot m = 30$ kg/s. Noting that $\dot m = \rho AV$, the inlet and outlet velocities of water are

$$V_1 = \frac{\dot m}{\rho A_1} = \frac{30\text{ kg/s}}{(1000\text{ kg/m}^3)(0.0150\text{ m}^2)} = 2.0\text{ m/s}$$

$$V_2 = \frac{\dot m}{\rho A_2} = \frac{30\text{ kg/s}}{(1000\text{ kg/m}^3)(0.0025\text{ m}^2)} = 12\text{ m/s}$$

Taking the center of the inlet cross section as the reference level ($z_1 = 0$) and noting that $P_2 = P_{atm}$, the Bernoulli equation for a streamline going through the center of the reducing elbow is expressed as

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \;\rightarrow\; P_1 - P_2 = \rho g\left(\frac{V_2^2 - V_1^2}{2g} + z_2 - z_1\right) \;\rightarrow\; P_{1,\text{ gage}} = \rho g\left(\frac{V_2^2 - V_1^2}{2g} + z_2\right)$$

or, $$P_{1,\text{ gage}} = (1000\text{ kg/m}^3)(9.81\text{ m/s}^2)\left(\frac{(12\text{ m/s})^2 - (2\text{ m/s})^2}{2(9.81\text{ m/s}^2)} + 0.4\right)\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 73.9\text{ kN/m}^2 = 73.9\text{ kPa}$$

The momentum equation for steady one-dimensional flow is $\sum \vec F = \sum_{\text{out}} \beta \dot m \vec V - \sum_{\text{in}} \beta \dot m \vec V$. We let the $x$- and $y$- components of the anchoring force of the elbow be $F_{Rx}$ and $F_{Rz}$, and assume them to be in the positive directions. We also use gage pressures to avoid dealing with the atmospheric pressure which acts on all surfaces. Then the momentum equations along the $x$ and $z$ axes become

$$F_{Rx} + P_{1,\text{gage}} A_1 = \beta \dot m V_2 \cos\theta - \beta \dot m V_1 \quad \text{and} \quad F_{Ry} - W = \beta \dot m V_2 \sin\theta$$

Solving for $F_{Rx}$ and $F_{Rz}$, and substituting the given values,

$$F_{Rx} = \beta \dot m (V_2 \cos\theta - V_1) - P_{1,\text{ gage}} A_1$$

$$= 1.03(30\text{ kg/s})[(12\cos 110° - 2)\text{ m/s}]\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) - (73.9\text{ kN/m}^2)(0.0150\text{ m}^2) = -1.297\text{ kN}$$

$$F_{Rz} = \beta \dot m V_2 \sin\theta + W = 1.03(30\text{ kg/s})(12\sin 110°\text{ m/s})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) + 0.4905\text{ kN} = 0.8389\text{ kN}$$

$$F_R = \sqrt{F_{Rx}^2 + F_{Rz}^2} = \sqrt{(-1.297)^2 + 0.8389^2} = \mathbf{1.54\text{ kN}}$$

and

$$\theta = \tan^{-1}\frac{F_{Rz}}{F_{Rx}} = \tan^{-1}\frac{0.8389}{-1.297} = \mathbf{-32.9°}$$

***Discussion*** Note that the magnitude of the anchoring force is 1.54 kN, and its line of action makes –32.9° from +$x$ direction. Negative value for $F_{Rx}$ indicates assumed direction is wrong, and should be reversed.

---

**6-27**

**Solution** Water accelerated by a nozzle strikes the back surface of a cart moving horizontally at a constant velocity. The braking force and the power wasted by the brakes are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The water splatters off the sides of the plate in all directions in the plane of the back surface. **3** The water jet is exposed to the atmosphere, and thus the pressure of the water jet and the splattered water is the atmospheric pressure which is disregarded since it acts on all surfaces. **4** Fiction during motion is negligible. **5** There is no acceleration of the cart. **7** The motions of the water jet and the cart are horizontal. **6** Jet flow is nearly uniform and thus the effect of the momentum-flux correction factor is negligible, $\beta \cong 1$.

***Analysis*** We take the cart as the control volume, and the direction of flow as the positive direction of *x* axis. The relative velocity between the cart and the jet is

$$V_r = V_{\text{jet}} - V_{\text{cart}} = 15 - 10 = 10 \text{ m/s}$$

5 m/s
15 m/s
Waterjet
$F_{Rx}$

Therefore, we can view the cart as being stationary and the jet moving with a velocity of 10 m/s. Noting that water leaves the nozzle at 15 m/s and the corresponding mass flow rate relative to nozzle exit is 25 kg/s, the mass flow rate of water striking the cart corresponding to a water jet velocity of 10 m/s relative to the cart is

$$\dot{m}_r = \frac{V_r}{V_{\text{jet}}}\dot{m}_{\text{jet}} = \frac{10 \text{ m/s}}{15 \text{ m/s}}(25 \text{ kg/s}) = 16.67 \text{ kg/s}$$

The momentum equation for steady one-dimensional flow in the *x* (flow) direction reduces in this case to

$$\sum \vec{F} = \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V} \quad \rightarrow \quad F_{Rx} = -\dot{m}_i V_i \quad \rightarrow \quad F_{\text{brake}} = -\dot{m}_r V_r$$

We note that the brake force acts in the opposite direction to flow, and we should not forget the negative sign for forces and velocities in the negative *x*-direction. Substituting the given values,

$$F_{\text{brake}} = -\dot{m}_r V_r = -(16.67 \text{ kg/s})(+10 \text{ m/s})\left(\frac{1 \text{ N}}{1 \text{ kg} \cdot \text{m/s}^2}\right) = -166.7 \text{ N} \cong \mathbf{-167 \text{ N}}$$

The negative sign indicates that the braking force acts in the opposite direction to motion, as expected. Noting that work is force times distance and the distance traveled by the cart per unit time is the cart velocity, the power wasted by the brakes is

$$\dot{W} = F_{\text{brake}} V_{\text{cart}} = (166.7 \text{ N})(5 \text{ m/s})\left(\frac{1 \text{ W}}{1 \text{ N} \cdot \text{m/s}}\right) = \mathbf{833 \text{ W}}$$

***Discussion*** Note that the power wasted is equivalent to the maximum power that can be generated as the cart velocity is maintained constant.

---

**6-28**

**Solution** Water accelerated by a nozzle strikes the back surface of a cart moving horizontally. The acceleration of the cart if the brakes fail is to be determined.

***Analysis*** The braking force was determined in previous problem to be 167 N. When the brakes fail, this force will propel the cart forward, and the acceleration will be

$$a = \frac{F}{m_{\text{cart}}} = \frac{167 \text{ N}}{300 \text{ kg}}\left(\frac{1 \text{ kg} \cdot \text{m/s}^2}{1 \text{ N}}\right) = \mathbf{0.556 \text{ m/s}^2}$$

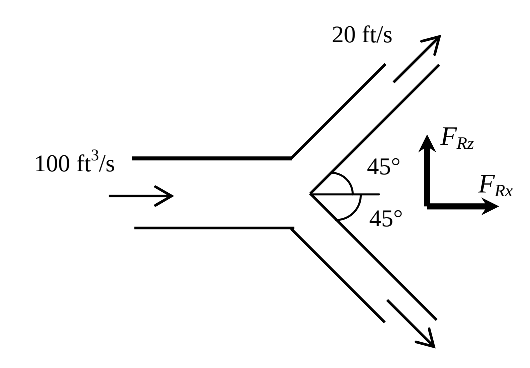
5 m/s
15 m/s
300 kg
Waterjet
$F_{Rx}$

***Discussion*** This is the acceleration at the moment the brakes fail. The acceleration will decrease as the relative velocity between the water jet and the cart (and thus the force) decreases.

---

**6-29E**

**Solution** A water jet hits a stationary splitter, such that half of the flow is diverted upward at 45°, and the other half is directed down. The force required to hold the splitter in place is to be determined.




***Assumptions*** **1** The flow is steady and incompressible. **2** The water jet is exposed to the atmosphere, and thus the pressure of the water jet before and after the split is the atmospheric pressure which is disregarded since it acts on all surfaces. **3** The gravitational effects are disregarded. **4** Jet flow is nearly uniform and thus the effect of the momentum-flux correction factor is negligible, $\beta \cong 1$.

***Properties*** We take the density of water to be 62.4 lbm/ft³.

***Analysis*** The mass flow rate of water jet is

$$\dot{m} = \rho\dot{V} = (62.4\ \text{lbm/ft}^3)(100\ \text{ft}^3/\text{s}) = 6240\ \text{lbm/s}$$

We take the splitting section of water jet, including the splitter as the control volume, and designate the entrance by 1 and the outlet of either arm by 2 (both arms have the same velocity and mass flow rate). We also designate the horizontal coordinate by $x$ with the direction of flow as being the positive direction and the vertical coordinate by $z$.

The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta\dot{m}\vec{V} - \sum_{\text{in}} \beta\dot{m}\vec{V}$. We let the $x$- and $y$-components of the anchoring force of the splitter be $F_{Rx}$ and $F_{Rz}$, and assume them to be in the positive directions. Noting that $V_2 = V_1 = V$ and $\dot{m}_2 = \frac{1}{2}\dot{m}$, the momentum equations along the $x$ and $z$ axes become

$$F_{Rx} = 2(\tfrac{1}{2}\dot{m})V_2 \cos\theta - \dot{m}V_1 = \dot{m}V(\cos\theta - 1)$$

$$F_{Rz} = \tfrac{1}{2}\dot{m}(+V_2 \sin\theta) + \tfrac{1}{2}\dot{m}(-V_2 \sin\theta) - 0 = 0$$

Substituting the given values,

$$F_{Rx} = (6240\ \text{lbm/s})(20\ \text{ft/s})(\cos 45° - 1)\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right) = --1135\ \text{lbf} \cong \mathbf{-1140\ lbf}$$

$$F_{Rz} = \mathbf{0}$$

The negative value for $F_{Rx}$ indicates the assumed direction is wrong, and should be reversed. Therefore, a force of 1140 lbf must be applied to the splitter in the opposite direction to flow to hold it in place. No holding force is necessary in the vertical direction. This can also be concluded from the symmetry.

***Discussion*** In reality, the gravitational effects will cause the upper stream to slow down and the lower stream to speed up after the split. But for short distances, these effects are indeed negligible.

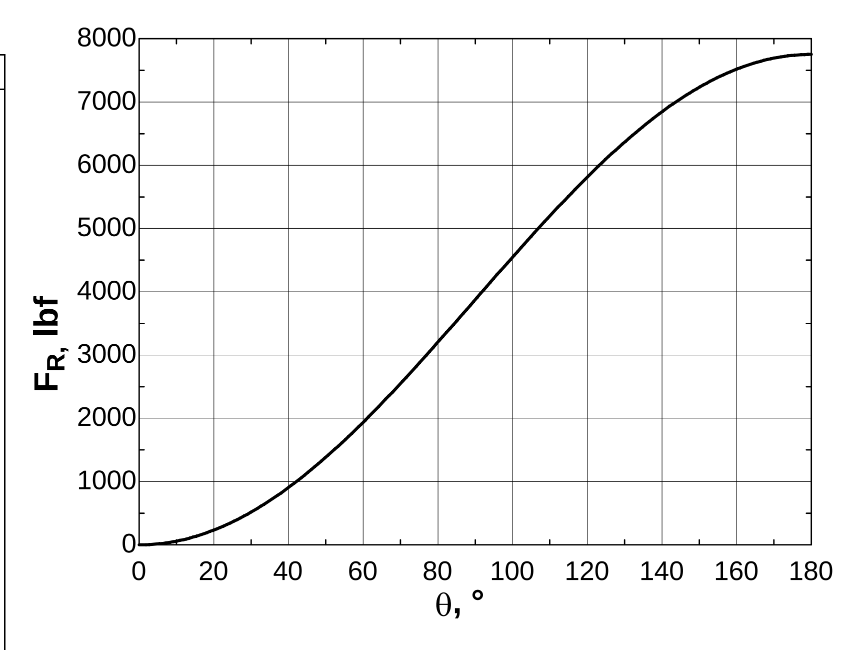

**6-30E** 

**Solution** The previous problem is reconsidered. The effect of splitter angle on the force exerted on the splitter as the half splitter angle varies from 0 to 180° in increments of 10° is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
g=32.2 "ft/s2"
rho=62.4 "lbm/ft3"
V_dot=100 "ft3/s"
V=20 "ft/s"
m_dot=rho*V_dot
F_R=-m_dot*V*(cos(theta)-1)/g "lbf"
```

| $\theta$, ° | $\dot{m}$, lbm/s | $F_R$, lbf |
|---|---|---|
| 0 | 6240 | 0 |
| 10 | 6240 | 59 |
| 20 | 6240 | 234 |
| 30 | 6240 | 519 |
| 40 | 6240 | 907 |
| 50 | 6240 | 1384 |
| 60 | 6240 | 1938 |
| 70 | 6240 | 2550 |
| 80 | 6240 | 3203 |
| 90 | 6240 | 3876 |
| 100 | 6240 | 4549 |
| 110 | 6240 | 5201 |
| 120 | 6240 | 5814 |
| 130 | 6240 | 6367 |
| 140 | 6240 | 6845 |
| 150 | 6240 | 7232 |
| 160 | 6240 | 7518 |
| 170 | 6240 | 7693 |
| 180 | 6240 | 7752 |




***Discussion*** The force rises from zero at $\theta = 0^o$ to a maximum at $\theta = 180^o$, as expected, but the relationship is not linear.

**6-31**

**Solution** A horizontal water jet impinges normally upon a vertical plate which is held on a frictionless track and is initially stationary. The initial acceleration of the plate, the time it takes to reach a certain velocity, and the velocity at a given time are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The water always splatters in the plane of the retreating plate. **3** The water jet is exposed to the atmosphere, and thus the pressure of the water jet and the splattered water is the atmospheric pressure which is disregarded since it acts on all surfaces. **4** The tract is nearly frictionless, and thus fiction during motion is negligible. **5** The motions of the water jet and the cart are horizontal. **6** The velocity of the jet relative to the plate remains constant, $V_r = V_{jet} = V$. **7** Jet flow is nearly uniform and thus the effect of the momentum-flux correction factor is negligible, $\beta \cong 1$.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** (*a*) We take the vertical plate on the frictionless track as the control volume, and the direction of flow as the positive direction of *x* axis. The mass flow rate of water in the jet is

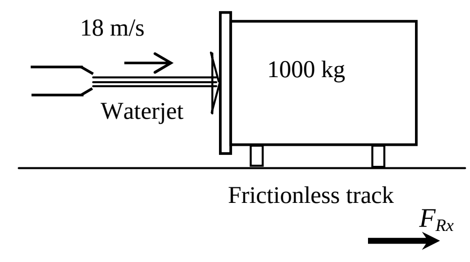
$$\dot{m} = \rho VA = (1000\ \text{kg/m}^3)(18\ \text{m/s})[\pi(0.05\ \text{m})^2/4] = 35.34\ \text{kg/s}$$

The momentum equation for steady one-dimensional flow in the *x* (flow) direction reduces in this case to

$$\sum \vec{F} = \sum_{\text{out}} \beta\dot{m}\vec{V} - \sum_{\text{in}} \beta\dot{m}\vec{V} \quad \rightarrow \quad F_{Rx} = -\dot{m}_i V_i \quad \rightarrow \quad F_{Rx} = -\dot{m}V$$

where $F_{Rx}$ is the reaction force required to hold the plate in place. When the plate is released, an equal and opposite impulse force acts on the plate, which is determined to

$$F_{\text{plate}} = -F_{Rx} = \dot{m}V = (35.34\ \text{kg/s})(18\ \text{m/s})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = 636\ \text{N}$$




Then the initial acceleration of the plate becomes

$$a = \frac{F_{\text{plate}}}{m_{\text{plate}}} = \frac{636\ \text{N}}{1000\ \text{kg}}\left(\frac{1\ \text{kg}\cdot\text{m/s}^2}{1\ \text{N}}\right) = \mathbf{0.636\ m/s^2}$$

This acceleration will remain constant during motion since the force acting on the plate remains constant.

(*b*) Noting that $a = dV/dt = \Delta V/\Delta t$ since the acceleration *a* is constant, the time it takes for the plate to reach a velocity of 9 m/s is

$$\Delta t = \frac{\Delta V_{\text{plate}}}{a} = \frac{(9-0)\ \text{m/s}}{0.636\ \text{m/s}^2} = \mathbf{14.2\ s}$$

(*c*) Noting that $a = dV/dt$ and thus $dV = adt$ and that the acceleration *a* is constant, the plate velocity in 20 s becomes

$$V_{\text{plate}} = V_{0,\ \text{plate}} + a\Delta t = 0 + (0.636\ \text{m/s}^2)(20\ \text{s}) = \mathbf{12.7\ m/s}$$

***Discussion*** The assumption that the relative velocity between the water jet and the plate remains constant is valid only for the initial moments of motion when the plate velocity is low unless the water jet is moving with the plate at the same velocity as the plate.

**6-32**

**Solution** A 90° reducer elbow deflects water downwards into a smaller diameter pipe. The resultant force exerted on the reducer by water is to be determined.

***Assumptions*** **1** The flow is steady, frictionless, one-dimensional, incompressible, and irrotational (so that the Bernoulli equation is applicable). **2** The weight of the elbow and the water in it is disregarded since the gravitational effects are negligible. **3** The momentum-flux correction factor for each inlet and outlet is given to be $\beta = 1.04$.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** We take the elbow as the control volume, and designate the entrance by 1 and the outlet by 2. We also designate the horizontal coordinate by $x$ (with the direction of flow as being the positive direction) and the vertical coordinate by $z$. The continuity equation for this one-inlet one-outlet steady flow system is $\dot{m}_1 = \dot{m}_2 = \dot{m} = 353.4$ kg/s. Noting that $\dot{m} = \rho AV$, the mass flow rate of water and its outlet velocity are



$$\dot{m} = \rho V_1 A_1 = \rho V_1 (\pi D_1^2/4) = (1000 \text{ kg/m}^3)(5 \text{ m/s})[\pi (0.3 \text{ m})^2/4] = 353.4 \text{ kg/s}$$

$$V_2 = \frac{\dot{m}}{\rho A_2} = \frac{\dot{m}}{\rho \pi D_2^2/4} = \frac{353.4 \text{ kg/s}}{(1000 \text{ kg/m}^3)[\pi (0.15 \text{ m})^2/4]} = 20 \text{ m/s}$$

The Bernoulli equation for a streamline going through the center of the reducing elbow is expressed as

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + z_2 \quad \rightarrow \quad P_2 = P_1 + \rho g\left(\frac{V_1^2 - V_2^2}{2g} + z_1 - z_2\right)$$

Substituting, the gage pressure at the outlet becomes

$$P_2 = (300 \text{ kPa}) + (1000 \text{ kg/m}^3)(9.81 \text{ m/s}^2)\left(\frac{(5 \text{ m/s})^2 - (20 \text{ m/s})^2}{2(9.81 \text{ m/s}^2)} + 0.5\right)\left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ kPa}}{1 \text{ kN/m}^2}\right) = 117.4 \text{ kPa}$$

The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V}$. We let the $x$- and $z$- components of the anchoring force of the elbow be $F_{Rx}$ and $F_{Rz}$, and assume them to be in the positive directions. Then the momentum equations along the $x$ and $z$ axes become

$$F_{Rx} + P_{1,\text{gage}} A_1 = 0 - \beta \dot{m} V_1$$
$$F_{Rz} - P_{2,\text{gage}} A_2 = \beta \dot{m}(-V_2) - 0$$

Note that we should not forget the negative sign for forces and velocities in the negative $x$ or $z$ direction. Solving for $F_{Rx}$ and $F_{Rz}$, and substituting the given values,

$$F_{Rx} = -\beta \dot{m} V_1 - P_{1,\text{gage}} A_1 = -1.04(353.4 \text{ kg/s})(5 \text{ m/s})\left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right) - (300 \text{ kN/m}^2)\frac{\pi (0.3 \text{ m})^2}{4} = -23.0 \text{ kN}$$

$$F_{Rz} = -\beta \dot{m} V_2 + P_{2,\text{gage}} A_1 = -1.04(353.4 \text{ kg/s})(20 \text{ m/s})\left(\frac{1 \text{ kN}}{1000 \text{ kg}\cdot\text{m/s}^2}\right) + (117.4 \text{ kN/m}^2)\frac{\pi (0.15 \text{ m})^2}{4} = -5.28 \text{ kN and}$$

$$F_R = \sqrt{F_{Rx}^2 + F_{Rz}^2} = \sqrt{(-23.0)^2 + (-5.28)^2} = \mathbf{23.6\ kN}$$

$$\theta = \tan^{-1}\frac{F_{Rz}}{F_{Rx}} = \tan^{-1}\frac{-5.28}{-23.0} = \mathbf{12.9°}$$

***Discussion*** The magnitude of the anchoring force is 23.6 kN, and its line of action makes 12.9° from +$x$ direction. Negative values for $F_{Rx}$ and $F_{Ry}$ indicate that the assumed directions are wrong, and should be reversed.

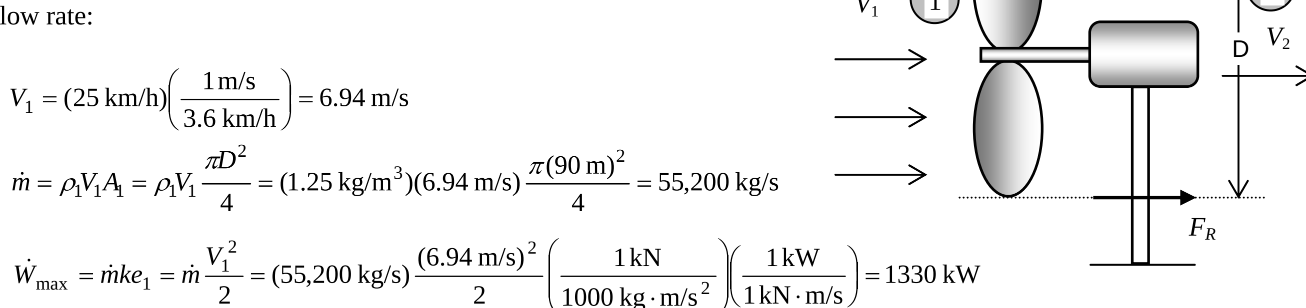
**6-33** **[***Also solved using EES on enclosed DVD***]**

**Solution** A wind turbine with a given span diameter and efficiency is subjected to steady winds. The power generated and the horizontal force on the supporting mast of the turbine are to be determined.

***Assumptions*** **1** The wind flow is steady and incompressible. **2** The efficiency of the turbine-generator is independent of wind speed. **3** The frictional effects are negligible, and thus none of the incoming kinetic energy is converted to thermal energy. **4** Wind flow is uniform and thus the momentum-flux correction factor is nearly unity, $\beta \cong 1$.

***Properties*** The density of air is given to be 1.25 kg/m$^3$.

***Analysis*** (*a*) The power potential of the wind is its kinetic energy, which is $V^2/2$ per unit mass, and $\dot{m}V^2/2$ for a given mass flow rate:




$$V_1 = (25\text{ km/h})\left(\frac{1\text{ m/s}}{3.6\text{ km/h}}\right) = 6.94\text{ m/s}$$

$$\dot{m} = \rho_1 V_1 A_1 = \rho_1 V_1 \frac{\pi D^2}{4} = (1.25\text{ kg/m}^3)(6.94\text{ m/s})\frac{\pi(90\text{ m})^2}{4} = 55{,}200\text{ kg/s}$$

$$\dot{W}_{max} = \dot{m}ke_1 = \dot{m}\frac{V_1^2}{2} = (55{,}200\text{ kg/s})\frac{(6.94\text{ m/s})^2}{2}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}}\right) = 1330\text{ kW}$$

Then the actual power produced becomes

$$\dot{W}_{act} = \eta_{\text{wind turbine}}\dot{W}_{max} = (0.32)(1330\text{ kW}) = \mathbf{426\text{ kW}}$$

(*b*) The frictional effects are assumed to be negligible, and thus the portion of incoming kinetic energy not converted to electric power leaves the wind turbine as outgoing kinetic energy. Therefore,

$$\dot{m}ke_2 = \dot{m}ke_1(1-\eta_{\text{wind turbine}}) \quad\rightarrow\quad \dot{m}\frac{V_2^2}{2} = \dot{m}\frac{V_1^2}{2}(1-\eta_{\text{wind turbine}})$$

or

$$V_2 = V_1\sqrt{1-\eta_{\text{wind turbine}}} = (6.94\text{ m/s})\sqrt{1-0.32} = 5.72\text{ m/s}$$

We choose the control volume around the wind turbine such that the wind is normal to the control surface at the inlet and the outlet, and the entire control surface is at the atmospheric pressure. The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{out}\beta\dot{m}\vec{V} - \sum_{in}\beta\dot{m}\vec{V}$. Writing it along the *x*-direction (without forgetting the negative sign for forces and velocities in the negative *x*-direction) and assuming the flow velocity through the turbine to be equal to the wind velocity give

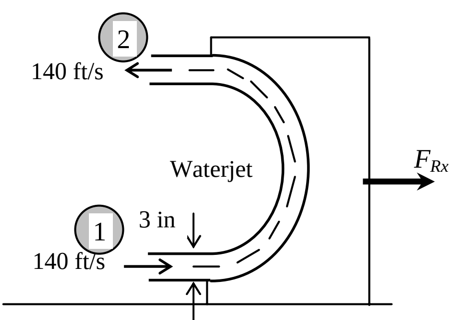
$$F_R = \dot{m}V_2 - \dot{m}V_1 = \dot{m}(V_2 - V_1) = (55{,}200\text{ kg/s})(5.72 - 6.94\text{ m/s})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = \mathbf{-67.3\text{ kN}}$$

The negative sign indicates that the reaction force acts in the negative *x* direction, as expected.

***Discussion*** This force acts on top of the tower where the wind turbine is installed, and the bending moment it generates at the bottom of the tower is obtained by multiplying this force by the tower height.

**6-34E**

**Solution** A horizontal water jet strikes a curved plate, which deflects the water back to its original direction. The force required to hold the plate against the water stream is to be determined.




***Assumptions*** **1** The flow is steady and incompressible. **2** The water jet is exposed to the atmosphere, and thus the pressure of the water jet and the splattered water is the atmospheric pressure, which is disregarded since it acts on all surfaces. **3** Friction between the plate and the surface it is on is negligible (or the friction force can be included in the required force to hold the plate). **4** There is no splashing of water or the deformation of the jet, and the reversed jet leaves horizontally at the same velocity and flow rate. **5** Jet flow is nearly uniform and thus the momentum-flux correction factor is nearly unity, $\beta \cong 1$.

***Properties*** We take the density of water to be 62.4 lbm/ft$^3$.

***Analysis*** We take the plate together with the curved water jet as the control volume, and designate the jet inlet by 1 and the outlet by 2. We also designate the horizontal coordinate by *x* (with the direction of incoming flow as being the positive direction). The continuity equation for this one-inlet one-outlet steady flow system is $\dot{m}_1 = \dot{m}_2 = \dot{m}$ where

$$\dot{m} = \rho VA = \rho V[\pi D^2 / 4] = (62.4 \text{ lbm/ft}^3)(140 \text{ ft/s})[\pi(3/12 \text{ ft})^2 / 4] = 428.8 \text{ lbm/s}$$

The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta \dot{m} \vec{V} - \sum_{\text{in}} \beta \dot{m} \vec{V}$. Letting the reaction force to hold the plate be $F_{Rx}$ and assuming it to be in the positive direction, the momentum equation along the *x* axis becomes

$$F_{Rx} = \dot{m}(-V_2) - \dot{m}(+V_1) = -2\dot{m}V$$

Substituting,

$$F_{Rx} = -2(428.8 \text{ lbm/s})(140 \text{ ft/s})\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm} \cdot \text{ft/s}^2}\right) = -3729 \text{ lbf} \cong \mathbf{-3730 \text{ lbf}}$$

Therefore, a force of 3730 lbf must be applied on the plate in the negative *x* direction to hold it in place.

***Discussion*** Note that a negative value for $F_{Rx}$ indicates the assumed direction is wrong (as expected), and should be reversed. Also, there is no need for an analysis in the vertical direction since the fluid streams are horizontal.

**6-35E**

**Solution** A horizontal water jet strikes a bent plate, which deflects the water by 135° from its original direction. The force required to hold the plate against the water stream is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The water jet is exposed to the atmosphere, and thus the pressure of the water jet and the splattered water is the atmospheric pressure, which is disregarded since it acts on all surfaces. **3** Frictional and gravitational effects are negligible. **4** There is no splattering of water or the deformation of the jet, and the reversed jet leaves horizontally at the same velocity and flow rate. **5** Jet flow is nearly uniform and thus the momentum-flux correction factor is nearly unity, $\beta \cong 1$.

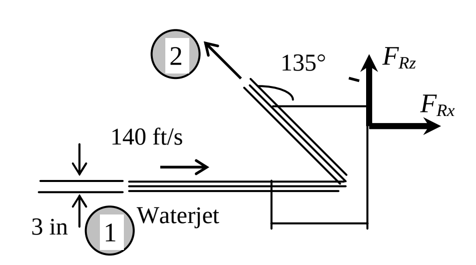
***Properties*** We take the density of water to be 62.4 lbm/ft³.

***Analysis*** We take the plate together with the curved water jet as the control volume, and designate the jet inlet by 1 and the outlet by 2. We also designate the horizontal coordinate by *x* (with the direction of incoming flow as being the positive direction), and the vertical coordinate by *z*. The continuity equation for this one-inlet one-outlet steady flow system is $\dot{m}_1 = \dot{m}_2 = \dot{m}$ where

$$\dot{m} = \rho VA = \rho V[\pi D^2/4] = (62.4\ \text{lbm/ft}^3)(140\ \text{ft/s})[\pi(3/12\ \text{ft})^2/4] = 428.8\ \text{lbm/s}$$

The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta\dot{m}\vec{V} - \sum_{\text{in}} \beta\dot{m}\vec{V}$. We let the *x*- and *z*- components of the anchoring force of the plate be $F_{Rx}$ and $F_{Rz}$, and assume them to be in the positive directions. Then the momentum equations along the *x* and *y* axes become

$$F_{Rx} = \dot{m}(-V_2)\cos 45° - \dot{m}(+V_1) = -\dot{m}V(1+\cos 45°)$$
$$F_{Rz} = \dot{m}(+V_2)\sin 45° = \dot{m}V\sin 45°$$




Substituting the given values,

$$F_{Rx} = -2(428.8\ \text{lbm/s})(140\ \text{ft/s})(1+\cos 45°)\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right)$$
$$= -6365\ \text{lbf}$$
$$F_{Rz} = (428.8\ \text{lbm/s})(140\ \text{ft/s})\sin 45°\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right) = 1318\ \text{lbf}$$

and $F_R = \sqrt{F_{Rx}^2 + F_{Rz}^2} = \sqrt{(-6365)^2 + 1318^2} = \mathbf{6500\ lbf,}$ $\quad \theta = \tan^{-1}\frac{F_{Ry}}{F_{Rx}} = \tan^{-1}\frac{1318}{-6365} = -11.7° = 168.3° \cong \mathbf{168°}$

***Discussion*** Note that the magnitude of the anchoring force is 6500 lbf, and its line of action is 168° from the positive *x* direction. Also, a negative value for $F_{Rx}$ indicates the assumed direction is wrong, and should be reversed.

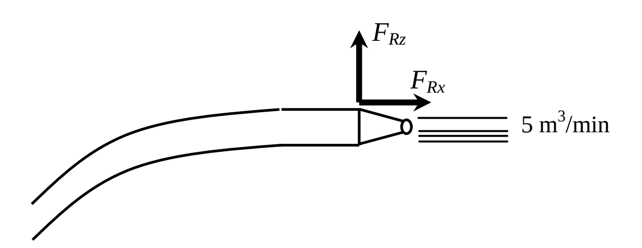
**6-36**

**Solution** Firemen are holding a nozzle at the end of a hose while trying to extinguish a fire. The average water outlet velocity and the resistance force required of the firemen to hold the nozzle are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The water jet is exposed to the atmosphere, and thus the pressure of the water jet is the atmospheric pressure, which is disregarded since it acts on all surfaces. **3** Gravitational effects and vertical forces are disregarded since the horizontal resistance force is to be determined. **5** Jet flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** (*a*) We take the nozzle and the horizontal portion of the hose as the system such that water enters the control volume vertically and outlets horizontally (this way the pressure force and the momentum flux at the inlet are in the vertical direction, with no contribution to the force balance in the horizontal direction), and designate the entrance by 1 and the outlet by 2. We also designate the horizontal coordinate by *x* (with the direction of flow as being the positive direction). The average outlet velocity and the mass flow rate of water are determined from




$$V = \frac{\dot{V}}{A} = \frac{\dot{V}}{\pi D^2/4} = \frac{5\ \text{m}^3/\text{min}}{\pi(0.06\ \text{m})^2/4} = 1768\ \text{m/min} = \mathbf{29.5\ m/s}$$

$$\dot{m} = \rho\dot{V} = (1000\ \text{kg/m}^3)(5\ \text{m}^3/\text{min}) = 5000\ \text{kg/min} = 83.3\ \text{kg/s}$$

(*b*) The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta\dot{m}\vec{V} - \sum_{\text{in}} \beta\dot{m}\vec{V}$. We let horizontal force applied by the firemen to the nozzle to hold it be $F_{Rx}$, and assume it to be in the positive *x* direction. Then the momentum equation along the *x* direction gives

$$F_{Rx} = \dot{m}V_e - 0 = \dot{m}V = (83.3\ \text{kg/s})(29.5\ \text{m/s})\left(\frac{1\ \text{N}}{1\text{kg}\cdot\text{m/s}^2}\right) = 2457\ \text{N} \cong \mathbf{2460\ N}$$

Therefore, the firemen must be able to resist a force of 2460 N to hold the nozzle in place.

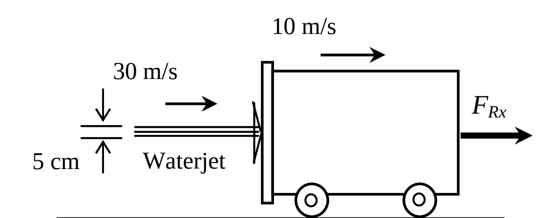
***Discussion*** The force of 2460 N is equivalent to the weight of about 250 kg. That is, holding the nozzle requires the strength of holding a weight of 250 kg, which cannot be done by a single person. This demonstrates why several firemen are used to hold a hose with a high flow rate.

**6-37**

**Solution** A horizontal jet of water with a given velocity strikes a flat plate that is moving in the same direction at a specified velocity. The force that the water stream exerts against the plate is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The water splatters in all directions in the plane of the plate. **3** The water jet is exposed to the atmosphere, and thus the pressure of the water jet and the splattered water is the atmospheric pressure, which is disregarded since it acts on all surfaces. **4** The vertical forces and momentum fluxes are not considered since they have no effect on the horizontal force exerted on the plate. **5** The velocity of the plate, and the velocity of the water jet relative to the plate, are constant. **6** Jet flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

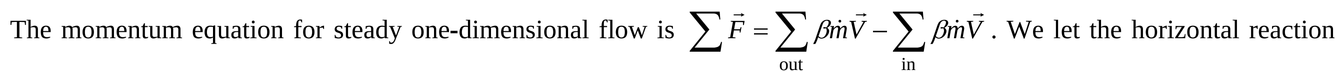



***Analysis*** We take the plate as the control volume, and the flow direction as the positive direction of $x$ axis. The relative velocity between the plate and the jet is

$$V_r = V_{\text{jet}} - V_{\text{plate}} = 30 - 10 = 20\text{ m/s}$$

Therefore, we can view the plate as being stationary and the jet to be moving with a velocity of 20 m/s. The mass flow rate of water relative to the plate [i.e., the flow rate at which water strikes the plate] is

$$\dot{m}_r = \rho V_r A = \rho V_r \frac{\pi D^2}{4} = (1000\text{ kg/m}^3)(20\text{ m/s})\frac{\pi(0.05\text{ m})^2}{4} = 39.27\text{ kg/s}$$

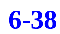


The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V}$. We let the horizontal reaction force applied to the plate in the negative $x$ direction to counteract the impulse of the water jet be $F_{Rx}$. Then the momentum equation along the $x$ direction gives

$$-F_{Rx} = 0 - \dot{m}V_i \quad \rightarrow \quad F_{Rx} = \dot{m}_r V_r = (39.27\text{ kg/s})(20\text{ m/s})\left(\frac{1\text{ N}}{1\text{kg}\cdot\text{m/s}^2}\right) = \mathbf{785\text{ N}}$$

Therefore, the water jet applies a force of 785 N on the plate in the direction of motion, and an equal and opposite force must be applied on the plate if its velocity is to remain constant.

***Discussion*** Note that we used the relative velocity in the determination of the mass flow rate of water in the momentum analysis since water will enter the control volume at this rate. (In the limiting case of the plate and the water jet moving at the same velocity, the mass flow rate of water relative to the plate will be zero since no water will be able to strike the plate).



**6-38** 

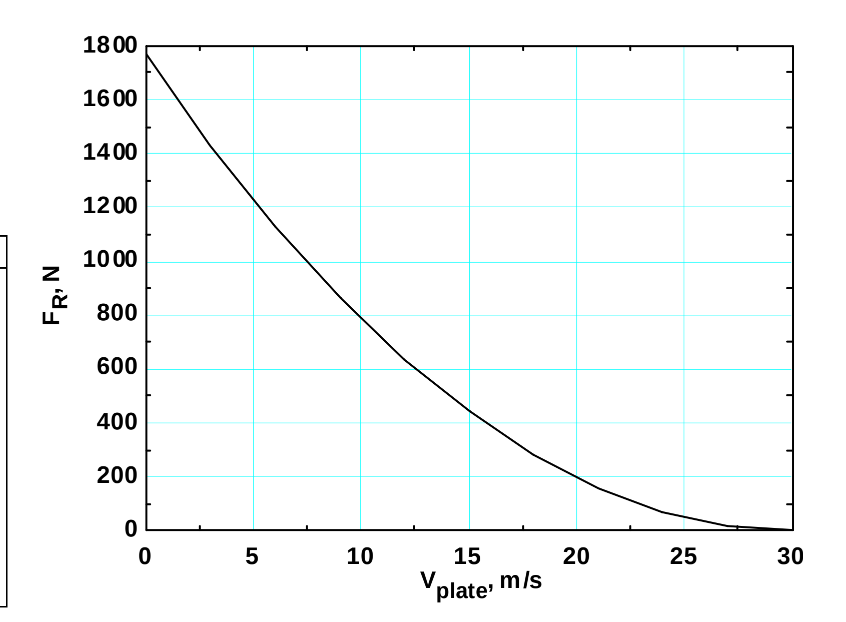
**Solution** The previous problem is reconsidered. The effect of the plate velocity on the force exerted on the plate as the plate velocity varies from 0 to 30 m/s in increments of 3 m/s is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
rho=1000 "kg/m3"
D=0.05 "m"
V_jet=30 "m/s"

Ac=pi*D^2/4
V_r=V_jet-V_plate
m_dot=rho*Ac*V_r
F_R=m_dot*V_r "N"
```

| $V_{plate}$, m/s | $V_r$, m/s | $F_R$, N |
|---|---|---|
| 0 | 30 | 1767 |
| 3 | 27 | 1431 |
| 6 | 24 | 1131 |
| 9 | 21 | 866 |
| 12 | 18 | 636 |
| 15 | 15 | 442 |
| 18 | 12 | 283 |
| 21 | 9 | 159 |
| 24 | 6 | 70.7 |
| 27 | 3 | 17.7 |
| 30 | 0 | 0 |



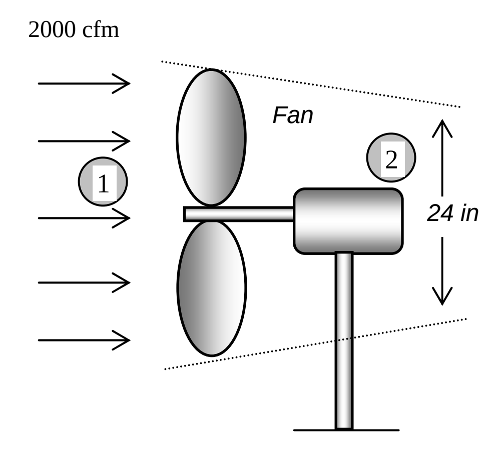

***Discussion*** When the plate velocity reaches 30 m/s, there is no relative motion between the jet and the plate; hence, there can be no force acting.

**6-39E**

**Solution** A fan moves air at sea level at a specified rate. The force required to hold the fan and the minimum power input required for the fan are to be determined.

***Assumptions*** **1** The flow of air is steady and incompressible. **2** Standard atmospheric conditions exist so that the pressure at sea level is 1 atm. **3** Air leaves the fan at a uniform velocity at atmospheric pressure. **4** Air approaches the fan through a large area at atmospheric pressure with negligible velocity. **5** The frictional effects are negligible, and thus the entire mechanical power input is converted to kinetic energy of air (no conversion to thermal energy through frictional effects). **6** Wind flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Properties*** The gas constant of air is $R$ = 0.3704 psi·ft$^3$/lbm·R. The standard atmospheric pressure at sea level is 1 atm = 14.7 psi.




***Analysis*** (*a*) We take the control volume to be a horizontal hyperbolic cylinder bounded by streamlines on the sides with air entering through the large cross-section (section 1) and the fan located at the narrow cross-section at the end (section 2), and let its centerline be the $x$ axis. The density, mass flow rate, and discharge velocity of air are

$$\rho = \frac{P}{RT} = \frac{14.7\text{ psi}}{(0.3704\text{ psi}\cdot\text{ft}^3/\text{lbm}\cdot\text{R})(530\text{ R})} = 0.0749\text{ lbm/ft}^3$$

$$\dot{m} = \rho\dot{V} = (0.0749\text{ lbm/ft}^3)(2000\text{ ft}^3/\text{min}) = 149.8\text{ lbm/min} = 2.50\text{ lbm/s}$$

$$V_2 = \frac{\dot{V}}{A_2} = \frac{\dot{V}}{\pi D_2^2/4} = \frac{2000\text{ ft}^3/\text{min}}{\pi(2\text{ ft})^2/4} = 636.6\text{ ft/min} = 10.6\text{ ft/s}$$

The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta\dot{m}\vec{V} - \sum_{\text{in}} \beta\dot{m}\vec{V}$. Letting the reaction force to hold the fan be $F_{Rx}$ and assuming it to be in the positive $x$ (i.e., the flow) direction, the momentum equation along the $x$ axis becomes

$$F_{Rx} = \dot{m}(V_2) - 0 = \dot{m}V = (2.50\text{ lbm/s})(10.6\text{ ft/s})\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = \mathbf{0.82\ lbf}$$

Therefore, a force of 0.82 lbf must be applied (through friction at the base, for example) to prevent the fan from moving in the horizontal direction under the influence of this force.

(*b*) Noting that $P_1 = P_2 = P_{\text{atm}}$ and $V_1 \cong 0$, the energy equation for the selected control volume reduces to

$$\dot{m}\left(\frac{P_1}{\rho} + \frac{V_1^2}{2} + gz_1\right) + \dot{W}_{\text{pump, u}} = \dot{m}\left(\frac{P_2}{\rho} + \frac{V_2^2}{2} + gz_2\right) + \dot{W}_{\text{turbine}} + \dot{E}_{\text{mech,loss}} \quad \rightarrow \quad \dot{W}_{\text{fan, u}} = \dot{m}\frac{V_2^2}{2}$$

Substituting,

$$\dot{W}_{\text{fan,u}} = \dot{m}\frac{V_2^2}{2} = (2.50\text{ lbm/s})\frac{(10.6\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1\text{ W}}{0.73756\text{ lbf}\cdot\text{ft/s}}\right) = \mathbf{5.91\ W}$$

Therefore, a useful mechanical power of 5.91 W must be supplied to air. This is the *minimum* required power input required for the fan.

***Discussion*** The actual power input to the fan will be larger than 5.91 W because of the fan inefficiency in converting mechanical power to kinetic energy.

**6-40**

**Solution** A helicopter hovers at sea level while being loaded. The volumetric air flow rate and the required power input during unloaded hover, and the rpm and the required power input during loaded hover are to be determined.

***Assumptions*** **1** The flow of air is steady and incompressible. **2** Air leaves the blades at a uniform velocity at atmospheric pressure. **3** Air approaches the blades from the top through a large area at atmospheric pressure with negligible velocity. **4** The frictional effects are negligible, and thus the entire mechanical power input is converted to kinetic energy of air (no conversion to thermal energy through frictional effects). **5** The change in air pressure with elevation is negligible because of the low density of air. **6** There is no acceleration of the helicopter, and thus the lift generated is equal to the total weight. **7** Air flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Properties*** The density of air is given to be 1.18 kg/m$^3$.

***Analysis*** (*a*) We take the control volume to be a vertical hyperbolic cylinder bounded by streamlines on the sides with air entering through the large cross-section (section 1) at the top and the fan located at the narrow cross-section at the bottom (section 2), and let its centerline be the *z* axis with upwards being the positive direction.

The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V}$. Noting that the only force acting on the control volume is the total weight *W* and it acts in the negative *z* direction, the momentum equation along the *z* axis gives

$$-W = \dot{m}(-V_2) - 0 \quad \rightarrow \quad W = \dot{m}V_2 = (\rho A V_2)V_2 = \rho A V_2^2 \quad \rightarrow \quad V_2 = \sqrt{\frac{W}{\rho A}}$$

where *A* is the blade span area,

$$A = \pi D^2/4 = \pi(15\text{ m})^2/4 = 176.7\text{ m}^2$$

Then the discharge velocity, volume flow rate, and the mass flow rate of air in the unloaded mode become

1

15 m

2

Sea level

Load
15,000 kg

$$V_{2,\text{unloaded}} = \sqrt{\frac{m_{\text{unloaded}}\, g}{\rho A}} = \sqrt{\frac{(10{,}000\text{ kg})(9.81\text{ m/s}^2)}{(1.18\text{ kg/m}^3)(176.7\text{ m}^2)}} = 21.7\text{ m/s}$$

$$\dot{V}_{\text{unloaded}} = AV_{2,\text{unloaded}} = \left(176.7\text{ m}^2\right)\left(21.7\text{ m/s}\right) = 3834\text{ m}^3\text{/s} \cong \mathbf{3830\ m^3/s}$$

$$\dot{m}_{\text{unloaded}} = \rho\dot{V}_{\text{unloaded}} = (1.18\text{ kg/m}^3)(3834\text{ m}^3\text{/s}) = 4524\text{ kg/s}$$



Noting that $P_1 = P_2 = P_{\text{atm}}$, $V_1 \cong 0$, the elevation effects are negligible, and the frictional effects are disregarded, the energy equation for the selected control volume reduces to

$$\dot{m}\left(\frac{P_1}{\rho} + \frac{V_1^2}{2} + gz_1\right) + \dot{W}_{\text{pump, u}} = \dot{m}\left(\frac{P_2}{\rho} + \frac{V_2^2}{2} + gz_2\right) + \dot{W}_{\text{turbine}} + \dot{E}_{\text{mech,loss}} \quad \rightarrow \quad \dot{W}_{\text{fan, u}} = \dot{m}\frac{V_2^2}{2}$$

Substituting,

$$\dot{W}_{\text{unloaded fan,u}} = \left(\dot{m}\frac{V_2^2}{2}\right)_{\text{unloaded}} = \left(4524\text{ kg/s}\right)\frac{(21.7\text{ m/s})^2}{2}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}}\right) = 1065\text{ kW} \cong \mathbf{1070\ kW}$$

(*b*) We now repeat the calculations for the *loaded* helicopter, whose mass is 10,000+15,000 = 25,000 kg:

$$V_{2,\text{loaded}} = \sqrt{\frac{m_{\text{loaded}}\, g}{\rho A}} = \sqrt{\frac{(25{,}000\text{ kg})(9.81\text{ m/s}^2)}{(1.18\text{ kg/m}^3)(176.7\text{ m}^2)}} = 34.3\text{ m/s}$$

$$\dot{m}_{\text{loaded}} = \rho\dot{V}_{\text{loaded}} = \rho AV_{2,\text{ loaded}} = (1.18\text{ kg/m}^3)(176.7\text{ m}^2)(34.3\text{ m/s}) = 7152\text{ kg/s}$$

$$\dot{W}_{\text{loaded fan,u}} = \left(\dot{m}\frac{V_2^2}{2}\right)_{\text{loaded}} = \left(7152\text{ kg/s}\right)\frac{(34.3\text{ m/s})^2}{2}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}}\right) = 4207\text{ kW} \cong \mathbf{4210\ kW}$$

Noting that the average flow velocity is proportional to the overhead blade rotational velocity, the rpm of the loaded helicopter blades becomes

$$V_2 = k\dot{n} \quad \rightarrow \quad \frac{V_{2,\text{loaded}}}{V_{2,\text{unloaded}}} = \frac{\dot{n}_{\text{loaded}}}{\dot{n}_{\text{unloaded}}} \quad \rightarrow \quad \dot{n}_{\text{loaded}} = \frac{V_{2,\text{loaded}}}{V_{2,\text{unloaded}}}\dot{n}_{\text{unloaded}} = \frac{34.3}{21.7}(400\text{ rpm}) = \mathbf{632\ rpm}$$

***Discussion*** The actual power input to the helicopter blades will be considerably larger than the calculated power input because of the fan inefficiency in converting mechanical power to kinetic energy.

**6-41**

**Solution** A helicopter hovers on top of a high mountain where the air density considerably lower than that at sea level. The blade rotational velocity to hover at the higher altitude and the percent increase in the required power input to hover at high altitude relative to that at sea level are to be determined.

***Assumptions*** **1** The flow of air is steady and incompressible. **2** The air leaves the blades at a uniform velocity at atmospheric pressure. **3** Air approaches the blades from the top through a large area at atmospheric pressure with negligible velocity. **4** The frictional effects are negligible, and thus the entire mechanical power input is converted to kinetic energy of air. **5** The change in air pressure with elevation while hovering at a given location is negligible because of the low density of air. **6** There is no acceleration of the helicopter, and thus the lift generated is equal to the total weight. **7** Air flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Properties*** The density of air is given to be 1.18 kg/m$^3$ at sea level, and 0.79 kg/m$^3$ on top of the mountain.

***Analysis*** (*a*) We take the control volume to be a vertical hyperbolic cylinder bounded by streamlines on the sides with air entering through the large cross-section (section 1) at the top and the fan located at the narrow cross-section at the bottom (section 2), and let its centerline be the *z* axis with upwards being the positive direction.

The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta \dot{m} \vec{V} - \sum_{\text{in}} \beta \dot{m} \vec{V}$. Noting that the only force acting on the control volume is the total weight *W* and it acts in the negative *z* direction, the momentum equation along the *z* axis gives

$$-W = \dot{m}(-V_2) - 0 \quad \rightarrow \quad W = \dot{m}V_2 = (\rho A V_2)V_2 = \rho A V_2^2 \quad \rightarrow \quad V_2 = \sqrt{\frac{W}{\rho A}}$$

where *A* is the blade span area. Then for a given weight *W*, the ratio of discharge velocities becomes

$$\frac{V_{2,\text{mountain}}}{V_{2,\text{sea}}} = \frac{\sqrt{W/\rho_{\text{mountain}} A}}{\sqrt{W/\rho_{\text{sea}} A}} = \sqrt{\frac{\rho_{\text{sea}}}{\rho_{\text{mountain}}}} = \sqrt{\frac{1.18 \text{ kg/m}^3}{0.79 \text{ kg/m}^3}} = 1.222$$

Noting that the average flow velocity is proportional to the overhead blade rotational velocity, the rpm of the helicopter blades on top of the mountain becomes

$$\dot{n} = kV_2 \quad \rightarrow \quad \frac{\dot{n}_{\text{mountain}}}{\dot{n}_{\text{sea}}} = \frac{V_{2,\text{mountain}}}{V_{2,\text{sea}}} \quad \rightarrow \quad \dot{n}_{\text{mountain}} = \frac{V_{2,\text{mountain}}}{V_{2,\text{sea}}} \dot{n}_{\text{sea}} = 1.222(400 \text{ rpm}) = \mathbf{489 \text{ rpm}}$$

Noting that $P_1 = P_2 = P_{\text{atm}}$, $V_1 \cong 0$, the elevation effect are negligible, and the frictional effects are disregarded, the energy equation for the selected control volume reduces to

$$\dot{m}\left(\frac{P_1}{\rho} + \frac{V_1^2}{2} + gz_1\right) + \dot{W}_{\text{pump, u}} = \dot{m}\left(\frac{P_2}{\rho} + \frac{V_2^2}{2} + gz_2\right) + \dot{W}_{\text{turbine}} + \dot{E}_{\text{mech,loss}} \quad \rightarrow \quad \dot{W}_{\text{fan, u}} = \dot{m}\frac{V_2^2}{2}$$

or $$\dot{W}_{\text{fan,u}} = \dot{m}\frac{V_2^2}{2} = \rho A V_2 \frac{V_2^2}{2} = \rho A \frac{V_2^3}{2} = \tfrac{1}{2}\rho A\left(\sqrt{\frac{W}{\rho A}}\right)^3 = \tfrac{1}{2}\rho A\left(\frac{W}{\rho A}\right)^{1.5} = \frac{W^{1.5}}{2\sqrt{\rho A}}$$

Then the ratio of the required power input on top of the mountain to that at sea level becomes

$$\frac{\dot{W}_{\text{mountain fan,u}}}{\dot{W}_{\text{sea fan,u}}} = \frac{0.5W^{1.5}/\sqrt{\rho_{\text{mountain}} A}}{0.5W^{1.5}/\sqrt{\rho_{\text{sea}} A}} = \sqrt{\frac{\rho_{\text{sea}}}{\rho_{\text{mountain}}}} = \sqrt{\frac{1.18 \text{ kg/m}^3}{0.79 \text{ kg/m}^3}} = 1.222$$

Therefore, the required power input will increase by **22.2%** on top of the mountain relative to the sea level.



***Discussion*** Note that both the rpm and the required power input to the helicopter are inversely proportional to the square root of air density. Therefore, more power is required at higher elevations for the helicopter to operate because air is less dense, and more air must be forced by the blades into the downdraft.

**6-42**

**Solution** The flow rate in a channel is controlled by a sluice gate by raising or lowering a vertical plate. A relation for the force acting on a sluice gate of width *w* for steady and uniform flow is to be developed.

***Assumptions*** **1** The flow is steady, incompressible, frictionless, and uniform (and thus the Bernoulli equation is applicable.) **2** Wall shear forces at surfaces are negligible. **3** The channel is exposed to the atmosphere, and thus the pressure at free surfaces is the atmospheric pressure. **4** The flow is horizontal. **5** Water flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Analysis*** We take point 1 at the free surface of the upstream flow before the gate and point 2 at the free surface of the downstream flow after the gate. We also take the bottom surface of the channel as the reference level so that the elevations of points 1 and 2 are $y_1$ and $y_2$, respectively. The application of the Bernoulli equation between points 1 and 2 gives

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+y_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+y_2 \quad \rightarrow \quad V_2^2-V_1^2=2g(y_1-y_2) \qquad (1)$$

The flow is assumed to be incompressible and thus the density is constant. Then the conservation of mass relation for this single stream steady flow device can be expressed as

$$\dot{V}_1=\dot{V}_2=\dot{V} \quad \rightarrow \quad A_1V_1=A_2V_2=\dot{V} \quad \rightarrow \quad V_1=\frac{\dot{V}}{A_1}=\frac{\dot{V}}{wy_1} \quad \text{and} \quad V_2=\frac{\dot{V}}{A_2}=\frac{\dot{V}}{wy_2} \qquad (2)$$

Substituting into Eq. (1),

$$\left(\frac{\dot{V}}{wy_2}\right)^2-\left(\frac{\dot{V}}{wy_1}\right)^2=2g(y_1-y_2) \rightarrow \dot{V}=w\sqrt{\frac{2g(y_1-y_2)}{1/y_2^2-1/y_1^2}} \quad \rightarrow \dot{V}=wy_2\sqrt{\frac{2g(y_1-y_2)}{1-y_2^2/y_1^2}} \qquad (3)$$

Substituting Eq. (3) into Eqs. (2) gives the following relations for velocities,

$$V_1=\frac{y_2}{y_1}\sqrt{\frac{2g(y_1-y_2)}{1-y_2^2/y_1^2}} \quad \text{and} \quad V_2=\sqrt{\frac{2g(y_1-y_2)}{1-y_2^2/y_1^2}} \qquad (4)$$

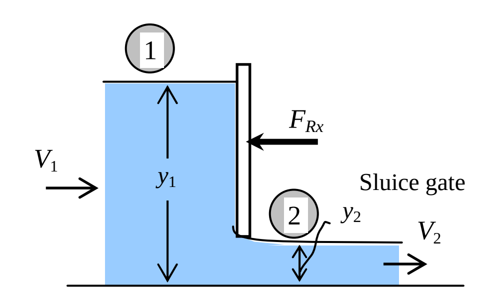
We choose the control volume as the water body surrounded by the vertical cross-sections of the upstream and downstream flows, free surfaces of water, the inner surface of the sluice gate, and the bottom surface of the channel. The momentum equation for steady one-dimensional flow is $\sum \vec{F}=\sum_{\text{out}} \beta\dot{m}\vec{V}-\sum_{\text{in}} \beta\dot{m}\vec{V}$. The force acting on the sluice gate $F_{Rx}$ is horizontal since the wall shear at the surfaces is negligible, and it is equal and opposite to the force applied on water by the sluice gate. Noting that the pressure force acting on a vertical surface is equal to the product of the pressure at the centroid of the surface and the surface area, the momentum equation along the *x* direction gives

$$-F_{Rx}+P_1A_1-P_2A_2=\dot{m}V_2-\dot{m}V_1 \quad \rightarrow \quad -F_{Rx}+\left(\rho g\frac{y_1}{2}\right)(wy_1)-\left(\rho g\frac{y_2}{2}\right)(wy_2)=\dot{m}(V_2-V_1)$$

Rearranging, the force acting on the sluice gate is determined to be

$$\boxed{F_{Rx}=\dot{m}(V_1-V_2)+\frac{w}{2}\rho g(y_1^2-y_2^2)} \qquad (5)$$

where $V_1$ and $V_2$ are given in Eq. (4).



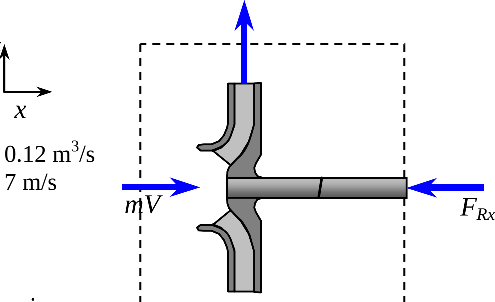

***Discussion*** Note that for $y_1 >> y_2$, Eq. (3) simplifies to $\dot{V}=y_2w\sqrt{2gy_1}$ or $V_2=\sqrt{2gy_1}$ which is the Toricelli equation for frictionless flow from a tank through a hole a distance $y_1$ below the free surface.

**6-43**

**Solution** Water enters a centrifugal pump axially at a specified rate and velocity, and leaves in the normal direction along the pump casing. The force acting on the shaft in the axial direction is to be determined.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Assumptions*** **1** The flow is steady and incompressible. **2** The forces acting on the piping system in the horizontal direction are negligible. **3** The atmospheric pressure is disregarded since it acts on all surfaces.




***Analysis*** We take the pump as the control volume, and the inlet direction of flow as the positive direction of *x* axis. The momentum equation for steady one-dimensional flow in the *x* (flow) direction reduces in this case to

$$\sum \vec{F} = \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V} \quad \rightarrow \quad -F_{Rx} = -\dot{m}V_i \quad \rightarrow \quad F_{Rx} = \dot{m}V_i = \rho \dot{V} V_i$$

Note that the reaction force acts in the opposite direction to flow, and we should not forget the negative sign for forces and velocities in the negative *x*-direction. Substituting the given values,

$$F_{\text{brake}} = (1000\ \text{kg/m}^3)(0.12\ \text{m}^3\text{/s})(7\ \text{m/s})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = \mathbf{840\ N}$$

***Discussion*** To find the total force acting on the shaft, we also need to do a force balance for the vertical direction, and find the vertical component of the reaction force.

---

## Angular Momentum Equation

**6-44C**

**Solution** We are to discuss how the angular momentum equation is obtained from the RTT.

***Analysis*** The *angular momentum equation* is obtained by **replacing *B* in the Reynolds transport theorem by the total angular momentum $\vec{H}_{sys}$, and *b* by the angular momentum per unit mass $\vec{r} \times \vec{V}$.**

***Discussion*** The RTT is a general equation that holds for any property *B*, either scalar or (as in this case) vector.

---

**6-45C**

**Solution** We are to express the angular momentum equation for a specific (restricted) control volume.

***Analysis*** The angular momentum equation in this case is expressed as $\boxed{I\vec{\alpha} = -\vec{r} \times \dot{m}\vec{V}}$ where $\vec{\alpha}$ is the angular acceleration of the control volume, and $\vec{r}$ is the vector from the axis of rotation to any point on the line of action of $\vec{F}$.

***Discussion*** This is a simplification of the more general angular momentum equation (many terms have dropped out).

---

**6-46C**

**Solution** We are to express the angular momentum equation in scalar form about a specified axis.

***Analysis*** The angular momentum equation about a given fixed axis in this case can be expressed in scalar form as $\boxed{\sum M = \sum_{\text{out}} r\dot{m}V - \sum_{\text{in}} r\dot{m}V}$ where *r* is the moment arm, *V* is the magnitude of the radial velocity, and $\dot{m}$ is the mass flow rate.

***Discussion*** This is a simplification of the more general angular momentum equation (many terms have dropped out).

---

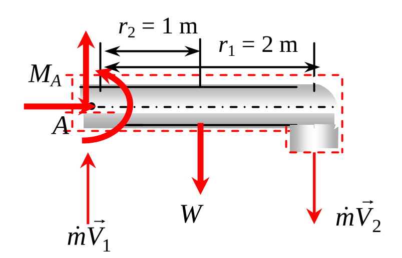
**6-47**

**Solution** Water is pumped through a piping section. The moment acting on the elbow for the cases of downward and upward discharge is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The water is discharged to the atmosphere, and thus the gage pressure at the outlet is zero. **3** Effects of water falling down during upward discharge is disregarded. **4** Pipe outlet diameter is small compared to the moment arm, and thus we use average values of radius and velocity at the outlet.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** We take the entire pipe as the control volume, and designate the inlet by 1 and the outlet by 2. We also take the $x$ and $y$ coordinates as shown. The control volume and the reference frame are fixed. The conservation of mass equation for this one-inlet one-outlet steady flow system is $\dot{m}_1 = \dot{m}_2 = \dot{m}$, and $V_1 = V_2 = V$ since $A_c$ = constant. The mass flow rate and the weight of the horizontal section of the pipe are




$$\dot{m} = \rho A_c V = (1000\ \text{kg/m}^3)[\pi(0.12\ \text{m})^2/4](4\ \text{m/s}) = 45.24\ \text{kg/s}$$

$$W = mg = (15\ \text{kg/m})(2\ \text{m})(9.81\ \text{m/s}^2)\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = 294.3\ \text{N/m}$$

($a$) **Downward discharge**: To determine the moment acting on the pipe at point $A$, we need to take the moment of all forces and momentum flows about that point. This is a steady and uniform flow problem, and all forces and momentum flows are in the same plane. Therefore, the angular momentum equation in this case can be expressed as $\sum M = \sum_{\text{out}} r\dot{m}V - \sum_{\text{in}} r\dot{m}V$ where $r$ is the moment arm, all moments in the counterclockwise direction are positive, and all in the clockwise direction are negative.

The free body diagram of the pipe section is given in the figure. Noting that the moments of all forces and momentum flows passing through point $A$ are zero, the only force that will yield a moment about point $A$ is the weight $W$ of the horizontal pipe section, and the only momentum flow that will yield a moment is the outlet stream (both are negative since both moments are in the clockwise direction). Then the angular momentum equation about point $A$ becomes

$$M_A - r_1 W = -r_2 \dot{m} V_2$$

Solving for $M_A$ and substituting,

$$M_A = r_1 W - r_2 \dot{m} V_2 = (1\ \text{m})(294.3\ \text{N}) - (2\ \text{m})(45.54\ \text{kg/s})(4\ \text{m/s})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = \mathbf{-70.0\ N\cdot m}$$

The negative sign indicates that the assumed direction for $M_A$ is wrong, and should be reversed. Therefore, a moment of 70 N·m acts at the stem of the pipe in the clockwise direction.

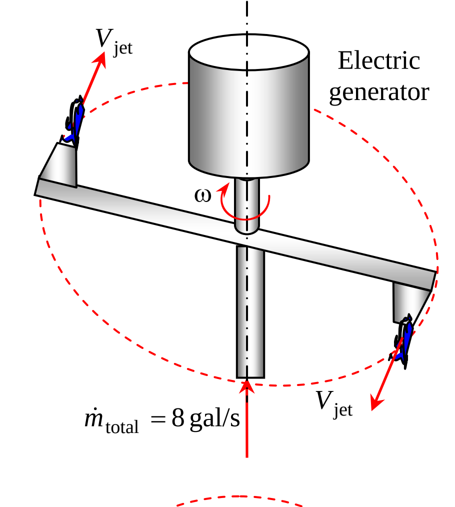
($b$) **Upward discharge**: The moment due to discharge stream is positive in this case, and the moment acting on the pipe at point $A$ is

$$M_A = r_1 W + r_2 \dot{m} V_2 = (1\ \text{m})(294.3\ \text{N}) + (2\ \text{m})(45.54\ \text{kg/s})(4\ \text{m/s})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = \mathbf{659\ N\cdot m}$$

***Discussion*** Note direction of discharge can make a big difference in the moments applied on a piping system. This problem also shows the importance of accounting for the moments of momentums of flow streams when performing evaluating the stresses in pipe materials at critical cross-sections.

**6-48E**

**Solution** A two-armed sprinkler is used to generate electric power. For a specified flow rate and rotational speed, the power produced is to be determined.




***Assumptions*** **1** The flow is cyclically steady (i.e., steady from a frame of reference rotating with the sprinkler head). **2** The water is discharged to the atmosphere, and thus the gage pressure at the nozzle outlet is zero. **3** Generator losses and air drag of rotating components are neglected. **4** The nozzle diameter is small compared to the moment arm, and thus we use average values of radius and velocity at the outlet.

***Properties*** We take the density of water to be 62.4 lbm/ft$^3$.

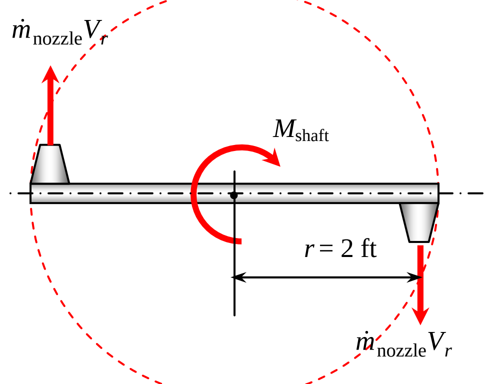
***Analysis*** We take the disk that encloses the sprinkler arms as the control volume, which is a stationary control volume. The conservation of mass equation for this steady flow system is $\dot{m}_1 = \dot{m}_2 = \dot{m}$. Noting that the two nozzles are identical, we have $\dot{m}_{nozzle} = \dot{m}/2$ or $\dot{V}_{nozzle} = \dot{V}_{total}/2$ since the density of water is constant. The average jet outlet velocity relative to the nozzle is

$$V_{jet} = \frac{\dot{V}_{nozzle}}{A_{jet}} = \frac{4 \text{ gal/s}}{[\pi(0.5/12 \text{ ft})^2/4]}\left(\frac{1 \text{ ft}^3}{7.480 \text{ gal}}\right) = 392.2 \text{ ft/s}$$

The angular and tangential velocities of the nozzles are

$$\omega = 2\pi\dot{n} = 2\pi(250 \text{ rev/min})\left(\frac{1 \text{ min}}{60 \text{ s}}\right) = 26.18 \text{ rad/s}$$

$$V_{nozzle} = r\omega = (2 \text{ ft})(26.18 \text{ rad/s}) = 52.36 \text{ ft/s}$$




The velocity of water jet relative to the control volume (or relative to a fixed location on earth) is

$$V_r = V_{jet} - V_{nozzle} = 392.2 - 52.36 = 339.8 \text{ ft/s}$$

The angular momentum equation can be expressed as $\sum M = \sum_{out} r\dot{m}V - \sum_{in} r\dot{m}V$ where all moments in the counterclockwise direction are positive, and all in the clockwise direction are negative. Then the angular momentum equation about the axis of rotation becomes

$$-M_{shaft} = -2r\dot{m}_{nozzle}V_r \quad \text{or} \quad M_{shaft} = r\dot{m}_{total}V_r$$

Substituting, the torque transmitted through the shaft is determined to be

$$M_{shaft} = r\dot{m}_{total}V_r = (2 \text{ ft})(66.74 \text{ lbm/s})(339.8 \text{ ft/s})\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm}\cdot\text{ft/s}^2}\right) = 1409 \text{ lbf}\cdot\text{ft}$$

since $\dot{m}_{total} = \rho\dot{V}_{total} = (62.4 \text{ lbm/ft}^3)(8/7.480 \text{ ft}^3\text{/s}) = 66.74 \text{ lbm/s}$. Then the power generated becomes

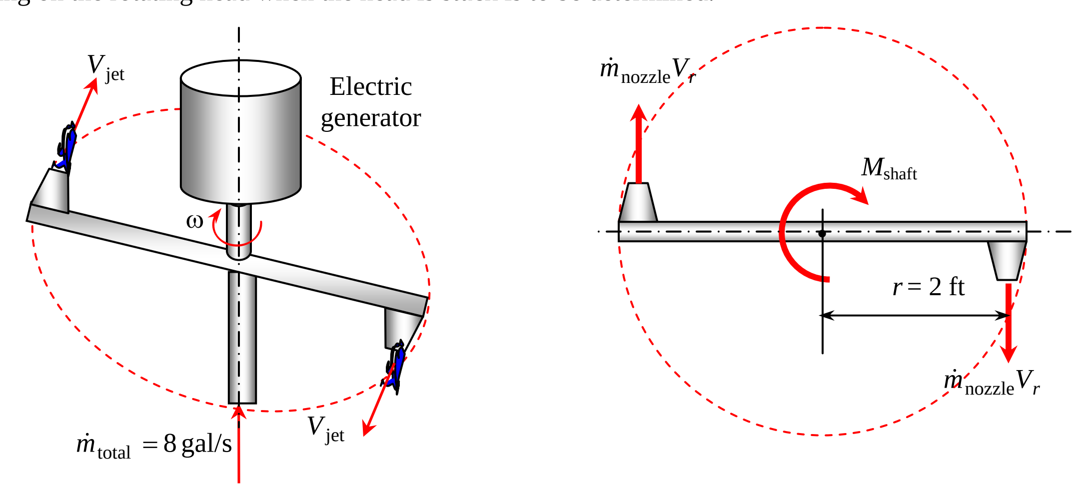
$$\dot{W} = 2\pi\dot{n}M_{shaft} = \omega M_{shaft} = (26.18 \text{ rad/s})(1409 \text{ lbf}\cdot\text{ft})\left(\frac{1 \text{ kW}}{737.56 \text{ lbf}\cdot\text{ft/s}}\right) = \mathbf{50.0 \text{ kW}}$$

Therefore, this sprinkler-type turbine has the potential to produce 50 kW of power.

***Discussion*** This is, of course, the maximum possible power. The actual power generated would be much smaller than this due to all the irreversible losses that we have ignored in this analysis.

**6-49E**

**Solution** A two-armed sprinkler is used to generate electric power. For a specified flow rate and rotational speed, the moment acting on the rotating head when the head is stuck is to be determined.




***Assumptions*** **1** The flow is uniform and steady. **2** The water is discharged to the atmosphere, and thus the gage pressure at the nozzle outlet is zero. **3** The nozzle diameter is small compared to the moment arm, and thus we use average values of radius and velocity at the outlet.

***Properties*** We take the density of water to be 62.4 lbm/ft³.

***Analysis*** We take the disk that encloses the sprinkler arms as the control volume, which is a stationary control volume. The conservation of mass equation for this steady flow system is $\dot{m}_1 = \dot{m}_2 = \dot{m}$. Noting that the two nozzles are identical, we have $\dot{m}_{nozzle} = \dot{m}/2$ or $\dot{V}_{nozzle} = \dot{V}_{total}/2$ since the density of water is constant. The average jet outlet velocity relative to the nozzle is

$$V_{jet} = \frac{\dot{V}_{nozzle}}{A_{jet}} = \frac{4 \text{ gal/s}}{[\pi(0.5/12 \text{ ft})^2/4]}\left(\frac{1 \text{ ft}^3}{7.480 \text{ gal}}\right) = 392.2 \text{ ft/s}$$

The angular momentum equation can be expressed as $\sum M = \sum_{out} r\dot{m}V - \sum_{in} r\dot{m}V$ where all moments in the counterclockwise direction are positive, and all in the clockwise direction are negative. Then the angular momentum equation about the axis of rotation becomes

$$-M_{shaft} = -2r\dot{m}_{nozzle}V_{jet} \quad \text{or} \quad M_{shaft} = r\dot{m}_{total}V_{jet}$$

Substituting, the torque transmitted through the shaft is determined to be

$$M_{shaft} = r\dot{m}_{total}V_{jet} = (2 \text{ ft})(66.74 \text{ lbm/s})(392.2 \text{ ft/s})\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm}\cdot\text{ft/s}^2}\right) = 1626 \text{ lbf}\cdot\text{ft} \cong \mathbf{1630 \text{ lbf}\cdot\text{ft}}$$

since $\dot{m}_{total} = \rho\dot{V}_{total} = (62.4 \text{ lbm/ft}^3)(8/7.480 \text{ ft}^3\text{/s}) = 66.74 \text{ lbm/s}$.

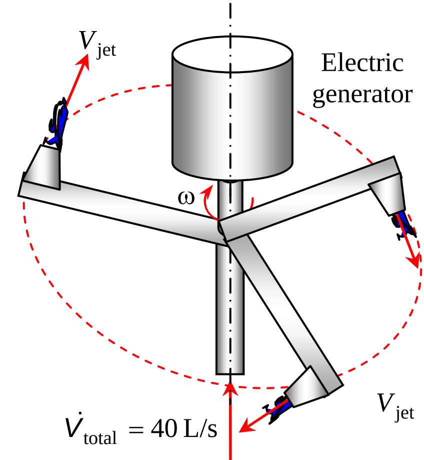
***Discussion*** When the sprinkler is stuck and thus the angular velocity is zero, the torque developed is maximum since $V_{nozzle} = 0$ and thus $V_r = V_{jet} = 392.2$ ft/s, giving $M_{shaft, max} = 1630$ lbf·ft. But the power generated is zero in this case since the shaft does not rotate.

**6-50**

**Solution** A three-armed sprinkler is used to water a garden. For a specified flow rate and resistance torque, the angular velocity of the sprinkler head is to be determined.

***Assumptions*** **1** The flow is uniform and cyclically steady (i.e., steady from a frame of reference rotating with the sprinkler head). **2** The water is discharged to the atmosphere, and thus the gage pressure at the nozzle outlet is zero. **3** Air drag of rotating components are neglected. **4** The nozzle diameter is small compared to the moment arm, and thus we use average values of radius and velocity at the outlet.

***Properties*** We take the density of water to be 1000 kg/m³ = 1 kg/L.



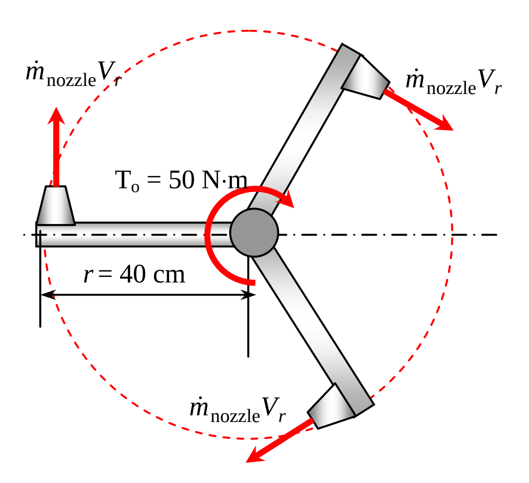

***Analysis*** We take the disk that encloses the sprinkler arms as the control volume, which is a stationary control volume. The conservation of mass equation for this steady flow system is $\dot{m}_1 = \dot{m}_2 = \dot{m}$. Noting that the three nozzles are identical, we have $\dot{m}_{nozzle} = \dot{m}/3$ or $\dot{V}_{nozzle} = \dot{V}_{total}/3$ since the density of water is constant. The average jet outlet velocity relative to the nozzle and the mass flow rate are

$$V_{jet} = \frac{\dot{V}_{nozzle}}{A_{jet}} = \frac{40\text{ L/s}}{3[\pi(0.012\text{ m})^2/4]}\left(\frac{1\text{ m}^3}{1000\text{ L}}\right) = 117.9\text{ m/s}$$

$$\dot{m}_{total} = \rho\dot{V}_{total} = (1\text{ kg/L})(40\text{ L/s}) = 40\text{ kg/s}$$

The angular momentum equation can be expressed as

$$\sum M = \sum_{out} r\dot{m}V - \sum_{in} r\dot{m}V$$




where all moments in the counterclockwise direction are positive, and all in the clockwise direction are negative. Then the angular momentum equation about the axis of rotation becomes

$$-T_0 = -3r\dot{m}_{nozzle}V_r \quad \text{or} \quad T_0 = r\dot{m}_{total}V_r$$

Solving for the relative velocity $V_r$ and substituting,

$$V_r = \frac{T_0}{r\dot{m}_{total}} = \frac{50\text{ N}\cdot\text{m}}{(0.40\text{ m})(40\text{ kg/s})}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right) = 3.1\text{ m/s}$$

Then the tangential and angular velocity of the nozzles become

$$V_{nozzle} = V_{jet} - V_r = 117.9 - 3.1 = 114.8\text{ m/s}$$

$$\omega = \frac{V_{nozzle}}{r} = \frac{114.8\text{ m/s}}{0.4\text{ m}} = \mathbf{287\ rad/s}$$

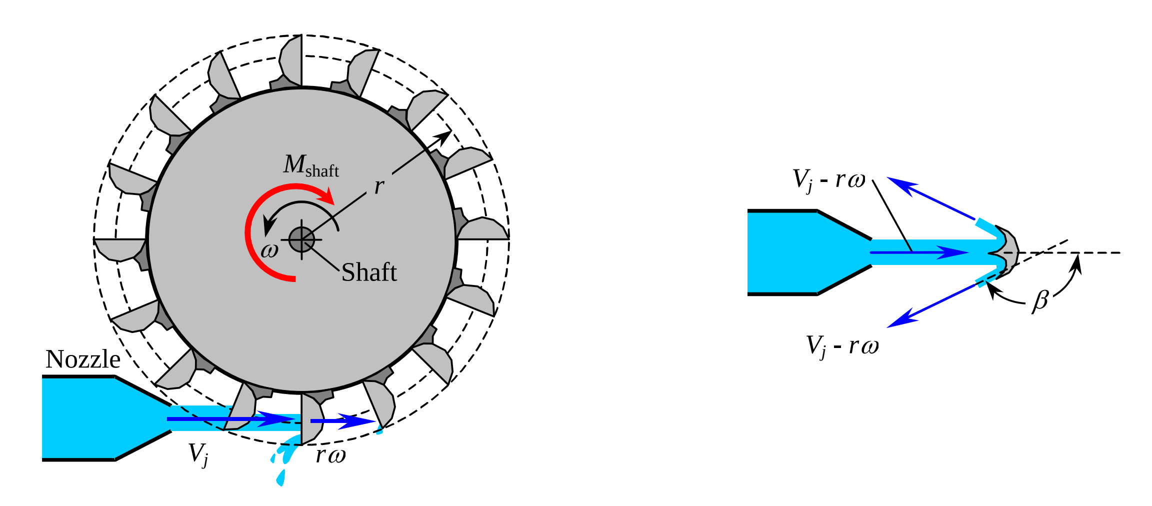
$$\dot{n} = \frac{\omega}{2\pi} = \frac{287\text{ rad/s}}{2\pi}\left(\frac{60\text{ s}}{1\text{ min}}\right) = 2741\text{ rpm} \cong \mathbf{2740\ rpm}$$

Therefore, this sprinkler will rotate at 2740 revolutions per minute (to three significant digits).

***Discussion*** The actual rotation rate will be somewhat lower than this due to air friction as the arms rotate.

**6-51**

**Solution** A Pelton wheel is considered for power generation in a hydroelectric power plant. A relation is to be obtained for power generation, and its numerical value is to be obtained.




***Assumptions*** **1** The flow is uniform and cyclically steady. **2** The water is discharged to the atmosphere, and thus the gage pressure at the nozzle outlet is zero. **3** Friction and losses due to air drag of rotating components are neglected. **4** The nozzle diameter is small compared to the moment arm, and thus we use average values of radius and velocity at the outlet.

***Properties*** We take the density of water to be 1000 kg/m$^3$ = 1 kg/L.

***Analysis*** The tangential velocity of buckets corresponding to an angular velocity of $\omega = 2\pi\dot{n}$ is $V_{\text{bucket}} = r\omega$. Then the relative velocity of the jet (relative to the bucket) becomes

$$V_r = V_j - V_{\text{bucket}} = V_j - r\omega$$

We take the imaginary disk that contains the Pelton wheel as the control volume. The inlet velocity of the fluid into this control volume is $V_r$, and the component of outlet velocity normal to the moment arm is $V_r\cos\beta$. The angular momentum equation can be expressed as $\sum M = \sum_{\text{out}} r\dot{m}V - \sum_{\text{in}} r\dot{m}V$ where all moments in the counterclockwise direction are positive, and all in the clockwise direction are negative. Then the angular momentum equation about the axis of rotation becomes

$$-M_{\text{shaft}} = r\dot{m}V_r\cos\beta - r\dot{m}V_r \quad \text{or} \quad M_{\text{shaft}} = r\dot{m}V_r(1-\cos\beta) = r\dot{m}(V_j - r\omega)(1-\cos\beta)$$

Noting that $\dot{W}_{\text{shaft}} = 2\pi\dot{n}M_{\text{shaft}} = \omega M_{\text{shaft}}$ and $\dot{m} = \rho\dot{V}$, the shaft power output of a Pelton turbine becomes

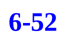
$$\boxed{\dot{W}_{\text{shaft}} = \rho\dot{V}r\omega(V_j - r\omega)(1-\cos\beta)}$$

which is the desired relation. For given values, the shaft power output is determined to be

$$\dot{W}_{\text{shaft}} = (1000\text{ kg/m}^3)(10\text{ m}^3\text{/s})(2\text{ m})(15.71\text{ rad/s})(50 - 2\times 15.71\text{ m/s})(1-\cos 160°)\left(\frac{1\text{ MW}}{10^6\text{ N}\cdot\text{m/s}}\right) = \mathbf{11.3\ MW}$$

where $\omega = 2\pi\dot{n} = 2\pi(150\text{ rev/min})\left(\frac{1\text{ min}}{60\text{ s}}\right) = 15.71\text{ rad/s}$

***Discussion*** The actual power will be somewhat lower than this due to air drag and friction. Note that this is the *shaft* power; the electrical power generated by the generator connected to the shaft is be lower due to generator inefficiencies.


**6-52**

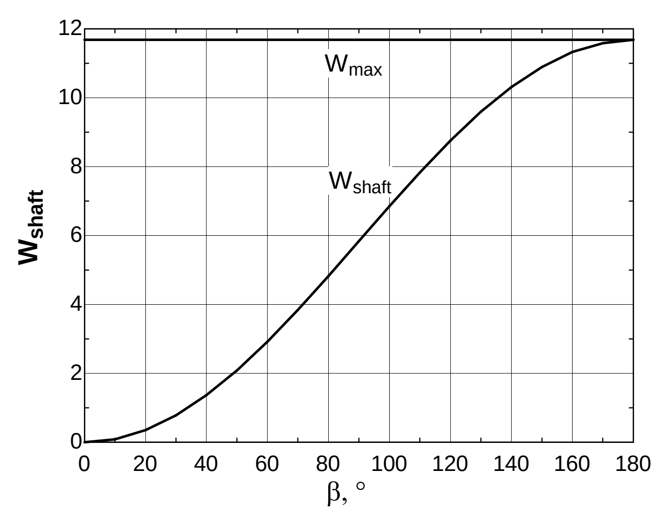


**Solution** The previous problem is reconsidered. The effect of $\beta$ on the power generation as $\beta$ varies from 0° to 180° is to be determined, and the fraction of power loss at 160° is to be assessed.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
rho=1000 "kg/m3"
r=2 "m"
V_dot=10 "m3/s"
V_jet=50 "m/s"
n_dot=150 "rpm"
omega=2*pi*n_dot/60
V_r=V_jet-r*omega
m_dot=rho*V_dot
W_dot_shaft=m_dot*omega*r*V_r*(1-cos(Beta))/1E6 "MW"
W_dot_max=m_dot*omega*r*V_r*2/1E6 "MW"
Efficiency=W_dot_shaft/W_dot_max
```

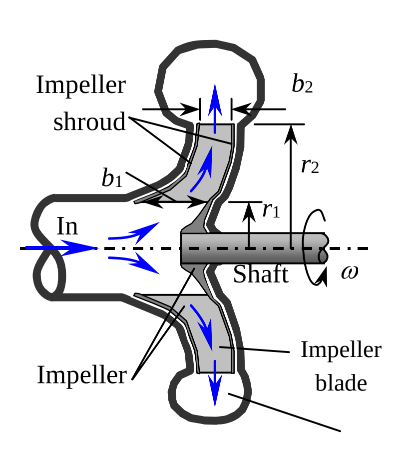
| Angle, $\beta$° | Max power, $\dot{W}_{max}$, MW | Actual power, $\dot{W}_{shaft}$, MW | Efficiency, $\eta$ |
|---|---|---|---|
| 0 | 11.7 | 0.00 | 0.000 |
| 10 | 11.7 | 0.09 | 0.008 |
| 20 | 11.7 | 0.35 | 0.030 |
| 30 | 11.7 | 0.78 | 0.067 |
| 40 | 11.7 | 1.37 | 0.117 |
| 50 | 11.7 | 2.09 | 0.179 |
| 60 | 11.7 | 2.92 | 0.250 |
| 70 | 11.7 | 3.84 | 0.329 |
| 80 | 11.7 | 4.82 | 0.413 |
| 90 | 11.7 | 5.84 | 0.500 |
| 100 | 11.7 | 6.85 | 0.587 |
| 110 | 11.7 | 7.84 | 0.671 |
| 120 | 11.7 | 8.76 | 0.750 |
| 130 | 11.7 | 9.59 | 0.821 |
| 140 | 11.7 | 10.31 | 0.883 |
| 150 | 11.7 | 10.89 | 0.933 |
| 160 | 11.7 | 11.32 | 0.970 |
| 170 | 11.7 | 11.59 | 0.992 |
| 180 | 11.7 | 11.68 | 1.000 |

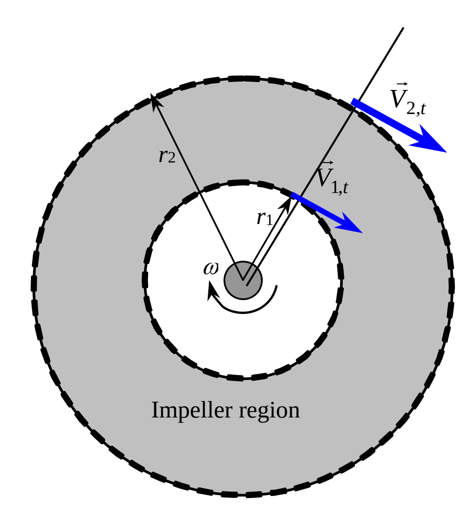



***Discussion*** The efficiency of a Pelton wheel for $\beta$=160° is 0.97. Therefore, at this angle, only 3% of the power is lost.

**6-53**

**Solution** A centrifugal blower is used to deliver atmospheric air. For a given angular speed and power input, the volume flow rate of air is to be determined.




***Assumptions*** **1** The flow is steady in the mean. **2** Irreversible losses are negligible. **3** The tangential components of air velocity at the inlet and the outlet are said to be equal to the impeller velocity at respective locations.

***Properties*** The gas constant of air is 0.287 kPa·m³/kg·K. The density of air at 20°C and 95 kPa is

$$\rho = \frac{P}{RT} = \frac{95 \text{ kPa}}{(0.287 \text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(293 \text{ K})} = 1.130 \text{ kg/m}^3$$

***Analysis*** In the idealized case of the tangential fluid velocity being equal to the blade angular velocity both at the inlet and the outlet, we have $V_{1,t} = \omega r_1$ and $V_{2,t} = \omega r_2$, and the torque is expressed as

$$\text{T}_{\text{shaft}} = \dot{m}(r_2 V_{2,t} - r_1 V_{1,t}) = \dot{m}\omega(r_2^2 - r_1^2) = \rho\dot{V}\omega(r_2^2 - r_1^2)$$

where the angular velocity is

$$\omega = 2\pi\dot{n} = 2\pi(800 \text{ rev/min})\left(\frac{1 \text{ min}}{60 \text{ s}}\right) = 83.78 \text{ rad/s}$$

Then the shaft power becomes

$$\dot{W}_{\text{shaft}} = \omega\text{T}_{\text{shaft}} = \rho\dot{V}\omega^2(r_2^2 - r_1^2)$$

Solving for $\dot{V}$ and substituting, the volume flow rate of air is determined to

$$\dot{V} = \frac{\dot{W}_{\text{shaft}}}{\rho\omega^2(r_2^2 - r_1^2)} = \frac{120 \text{ N}\cdot\text{m/s}}{(1.130 \text{ kg/m}^3)(83.78 \text{ rad/s})^2[(0.30 \text{ m})^2 - (0.15 \text{ m})^2]}\left(\frac{1 \text{ kg}\cdot\text{m/s}^2}{1 \text{ N}}\right) = \mathbf{0.224 \ m^3/s}$$

The normal velocity components at the inlet and the outlet are

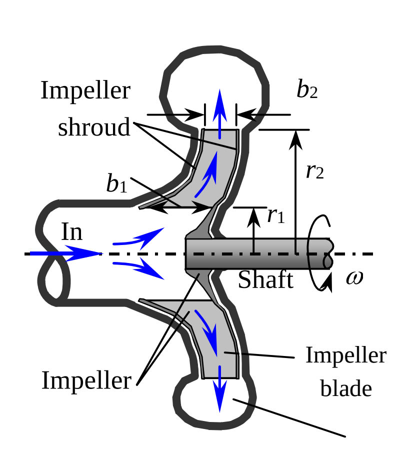
$$V_{1,n} = \frac{\dot{V}}{2\pi r_1 b_1} = \frac{0.224 \text{ m}^3/\text{s}}{2\pi(0.15 \text{ m})(0.061 \text{ m})} = \mathbf{3.90 \ m/s}$$

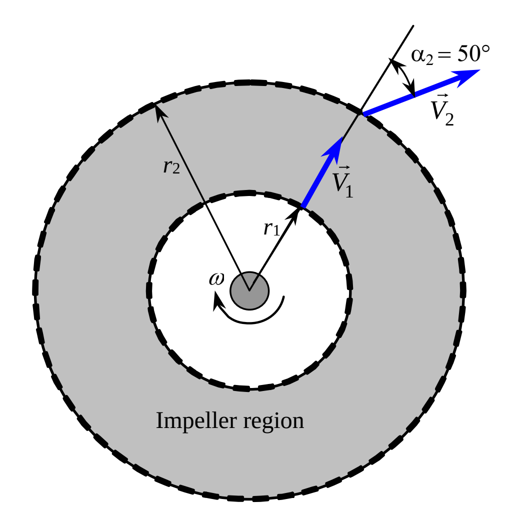
$$V_{2,n} = \frac{\dot{V}}{2\pi r_2 b_2} = \frac{0.224 \text{ m}^3/\text{s}}{2\pi(0.30 \text{ m})(0.034 \text{ m})} = \mathbf{3.50 \ m/s}$$

***Discussion*** Note that the irreversible losses are not considered in this analysis. In reality, the flow rate and the normal components of velocities will be smaller.

**6-54**

**Solution** A centrifugal blower is used to deliver atmospheric air at a specified rate and angular speed. The minimum power consumption of the blower is to be determined.




***Assumptions*** **1** The flow is steady in the mean. **2** Irreversible losses are negligible.

***Properties*** The density of air is given to be 1.25 kg/m$^3$.

***Analysis*** We take the impeller region as the control volume. The normal velocity components at the inlet and the outlet are

$$V_{1,n} = \frac{\dot{V}}{2\pi r_1 b_1} = \frac{0.70 \text{ m}^3\text{/s}}{2\pi(0.20 \text{ m})(0.082 \text{ m})} = 6.793 \text{ m/s}$$

$$V_{2,n} = \frac{\dot{V}}{2\pi r_2 b_2} = \frac{0.70 \text{ m}^3\text{/s}}{2\pi(0.45 \text{ m})(0.056 \text{ m})} = 4.421 \text{ m/s}$$

The tangential components of absolute velocity are:

$\alpha_1 = 0°$: $\quad V_{1,t} = V_{1,n} \tan\alpha_1 = 0$

$\alpha_2 = 60°$: $\quad V_{2,t} = V_{2,n} \tan\alpha_1 = (4.421 \text{ m/s}) \tan 50° = 5.269 \text{ m/s}$

The angular velocity of the propeller is

$$\omega = 2\pi\dot{n} = 2\pi(700 \text{ rev/min})\left(\frac{1 \text{ min}}{60 \text{ s}}\right) = 73.30 \text{ rad/s}$$

$$\dot{m} = \rho\dot{V} = (1.25 \text{ kg/m}^3)(0.7 \text{ m}^3\text{/s}) = 0.875 \text{ kg/s}$$

Normal velocity components $V_{1,n}$ and $V_{2,n}$ as well pressure acting on the inner and outer circumferential areas pass through the shaft center, and thus they do not contribute to torque. Only the tangential velocity components contribute to torque, and the application of the angular momentum equation gives

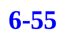
$$\text{T}_{\text{shaft}} = \dot{m}(r_2 V_{2,t} - r_1 V_{1,t}) = (0.875 \text{ kg/s})[(0.45 \text{ m})(5.269 \text{ m/s}) - 0]\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = 2.075 \text{ N}\cdot\text{m}$$

Then the shaft power becomes

$$\dot{W} = \omega \text{T}_{\text{shaft}} = (73.30 \text{ rad/s})(2.075 \text{ N}\cdot\text{m})\left(\frac{1 \text{ W}}{1 \text{ N}\cdot\text{m/s}}\right) = \mathbf{152\ W}$$

***Discussion*** The actual required shaft power is greater than this, due to the friction and other irreversibilities that we have neglected in our analysis. Nevertheless, this is a good first approximation.

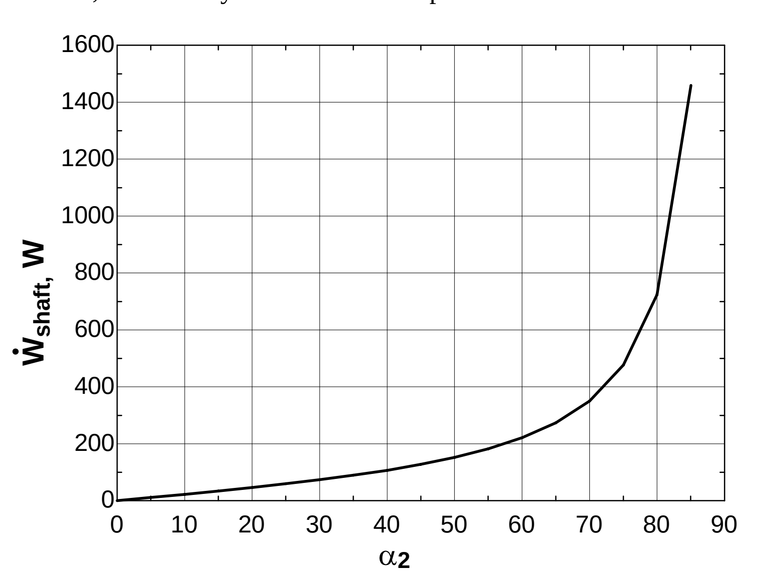
---


**6-55** 

**Solution** The previous problem is reconsidered. The effect of discharge angle $\alpha_2$ on the minimum power input requirements as $\alpha_2$ varies from 0° to 85° in increments of 5° is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
rho=1.25 "kg/m3"
r1=0.20 "m"
b1=0.082 "m"
r2=0.45 "m"
b2=0.056 "m"
V_dot=0.70 "m3/s"
V1n=V_dot/(2*pi*r1*b1) "m/s"
V2n=V_dot/(2*pi*r2*b2) "m/s"
Alpha1=0
V1t=V1n*tan(Alpha1) "m/s"
V2t=V2n*tan(Alpha2) "m/s"
n_dot=700 "rpm"
omega=2*pi*n_dot/60 "rad/s"
m_dot=rho*V_dot "kg/s"
T_shaft=m_dot*(r2*V2t-r1*V1t) "Nm"
W_dot_shaft=omega*T_shaft "W"
```

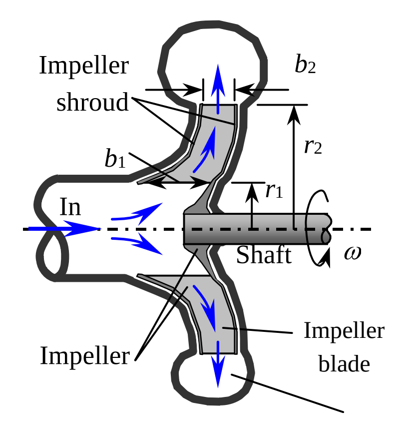


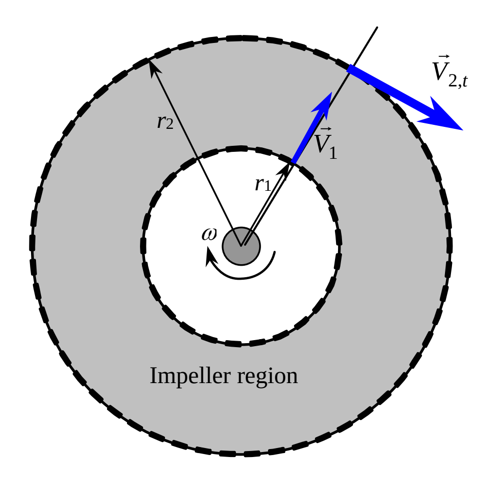

| Angle, $\alpha_2$° | $V_{2,t}$, m/s | Torque, $T_{shaft}$, Nm | Shaft power, $\dot{W}_{shaft}$, W |
|---|---|---|---|
| 0 | 0.00 | 0.00 | 0 |
| 5 | 0.39 | 0.15 | 11 |
| 10 | 0.78 | 0.31 | 23 |
| 15 | 1.18 | 0.47 | 34 |
| 20 | 1.61 | 0.63 | 46 |
| 25 | 2.06 | 0.81 | 60 |
| 30 | 2.55 | 1.01 | 74 |
| 35 | 3.10 | 1.22 | 89 |
| 40 | 3.71 | 1.46 | 107 |
| 45 | 4.42 | 1.74 | 128 |
| 50 | 5.27 | 2.07 | 152 |
| 55 | 6.31 | 2.49 | 182 |
| 60 | 7.66 | 3.02 | 221 |
| 65 | 9.48 | 3.73 | 274 |
| 70 | 12.15 | 4.78 | 351 |
| 75 | 16.50 | 6.50 | 476 |
| 80 | 25.07 | 9.87 | 724 |
| 85 | 50.53 | 19.90 | 1459 |

***Discussion*** When $\alpha_2$ = 0, the shaft power is also zero as expected, since there is no turning at all. As $\alpha_2$ approaches 90°, the required shaft power rises rapidly towards infinity. We can never reach $\alpha_2$ = 90° because this would mean zero flow normal to the outlet, which is impossible.

**6-56E**

**Solution** Water enters the impeller of a centrifugal pump radially at a specified flow rate and angular speed. The torque applied to the impeller is to be determined.




***Assumptions*** **1** The flow is steady in the mean. **2** Irreversible losses are negligible.

***Properties*** We take the density of water to be 62.4 lbm/ft$^3$.

***Analysis*** Water enters the impeller normally, and thus $V_{1,t} = 0$. The tangential component of fluid velocity at the outlet is given to be $V_{2,t} = 180 \text{ ft/s}$. The inlet radius $r_1$ is unknown, but the outlet radius is given to be $r_2$ = 1 ft. The angular velocity of the propeller is

$$\omega = 2\pi\dot{n} = 2\pi(500 \text{ rev/min})\left(\frac{1 \text{ min}}{60 \text{ s}}\right) = 52.36 \text{ rad/s}$$

The mass flow rate is

$$\dot{m} = \rho\dot{V} = (62.4 \text{ lbm/ft}^3)(80/60 \text{ ft}^3\text{/s}) = 83.2 \text{ lbm/s}$$

Only the tangential velocity components contribute to torque, and the application of the angular momentum equation gives

$$\text{T}_{\text{shaft}} = \dot{m}(r_2 V_{2,t} - r_1 V_{1,t}) = (83.2 \text{ lbm/s})[(1 \text{ ft})(180 \text{ ft/s}) - 0]\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm} \cdot \text{ft/s}^2}\right) = \mathbf{465\ lbf \cdot ft}$$

***Discussion*** This shaft power input corresponding to this torque is

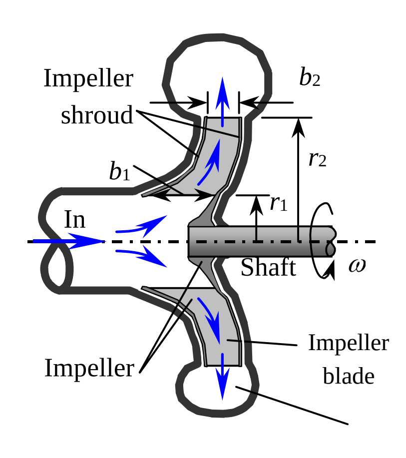
$$\dot{W} = 2\pi\dot{n}T_{\text{shaft}} = \omega T_{\text{shaft}} = (52.36 \text{ rad/s})(465 \text{ lbf} \cdot \text{ft})\left(\frac{1 \text{ kW}}{737.56 \text{ lbf} \cdot \text{ft/s}}\right) = \mathbf{33.0\ kW}$$

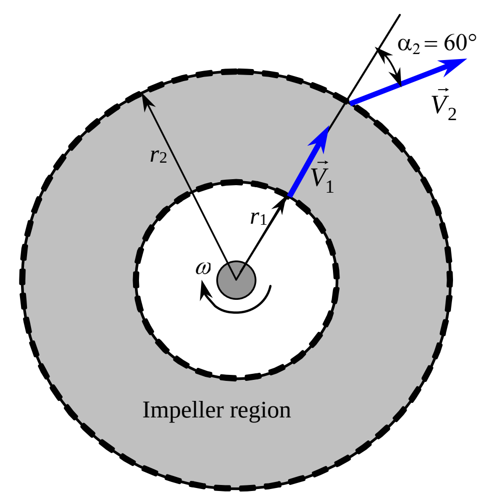
Therefore, the minimum power input to this pump should be 33 kW.

---

**6-57**

**Solution** A centrifugal pump is used to supply water at a specified rate and angular speed. The minimum power consumption of the pump is to be determined.




***Assumptions*** **1** The flow is steady in the mean. **2** Irreversible losses are negligible.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** We take the impeller region as the control volume. The normal velocity components at the inlet and the outlet are

$$V_{1,n} = \frac{\dot{V}}{2\pi r_1 b_1} = \frac{0.15\ \text{m}^3/\text{s}}{2\pi(0.13\ \text{m})(0.080\ \text{m})} = 2.296\ \text{m/s}$$

$$V_{2,n} = \frac{\dot{V}}{2\pi r_2 b_2} = \frac{0.15\ \text{m}^3/\text{s}}{2\pi(0.30\ \text{m})(0.035\ \text{m})} = 2.274\ \text{m/s}$$

The tangential components of absolute velocity are:

$\alpha_1 = 0°$: $V_{1,t} = V_{1,n}\tan\alpha_1 = 0$

$\alpha_2 = 60°$: $V_{2,t} = V_{2,n}\tan\alpha_1 = (2.274\ \text{m/s})\tan 60° = 3.938\ \text{m/s}$

The angular velocity of the propeller is

$$\omega = 2\pi\dot{n} = 2\pi(1200\ \text{rev/min})\left(\frac{1\ \text{min}}{60\ \text{s}}\right) = 125.7\ \text{rad/s}$$

$$\dot{m} = \rho\dot{V} = (1000\ \text{kg/m}^3)(0.15\ \text{m}^3/\text{s}) = 150\ \text{kg/s}$$

Normal velocity components $V_{1,n}$ and $V_{2,n}$ as well pressure acting on the inner and outer circumferential areas pass through the shaft center, and thus they do not contribute to torque. Only the tangential velocity components contribute to torque, and the application of the angular momentum equation gives

$$\text{T}_{\text{shaft}} = \dot{m}(r_2V_{2,t} - r_1V_{1,t}) = (150\ \text{kg/s})[(0.30\ \text{m})(3.938\ \text{m/s}) - 0]\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) = 177.2\ \text{kN}\cdot\text{m}$$

Then the shaft power becomes

$$\dot{W} = \omega\text{T}_{\text{shaft}} = (125.7\ \text{rad/s})(177.2\ \text{kN}\cdot\text{m})\left(\frac{1\ \text{kW}}{1\ \text{kN}\cdot\text{m/s}}\right) = \mathbf{22.3\ kW}$$

***Discussion*** Note that the irreversible losses are not considered in analysis. In reality, the required power input will be larger.

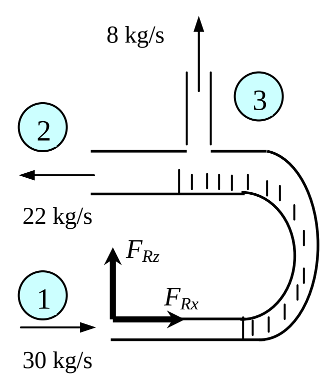
## Review Problems

**6-58**

**Solution** Water is flowing into and discharging from a pipe U-section with a secondary discharge section normal to return flow. Net *x*- and *z*- forces at the two flanges that connect the pipes are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The weight of the U-turn and the water in it is negligible. **4** The momentum-flux correction factor for each inlet and outlet is given to be $\beta = 1.03$.

***Properties*** We take the density of water to be 1000 kg/m$^3$.




***Analysis*** The flow velocities of the 3 streams are

$$V_1 = \frac{\dot{m}_1}{\rho A_1} = \frac{\dot{m}_1}{\rho(\pi D_1^2/4)} = \frac{30\ \text{kg/s}}{(1000\ \text{kg/m}^3)[\pi(0.05\ \text{m})^2/4]} = 15.3\ \text{m/s}$$

$$V_2 = \frac{\dot{m}_2}{\rho A_2} = \frac{\dot{m}_2}{\rho(\pi D_2^2/4)} = \frac{22\ \text{kg/s}}{(1000\ \text{kg/m}^3)[\pi(0.10\ \text{m})^2/4]} = 2.80\ \text{m/s}$$

$$V_3 = \frac{\dot{m}_3}{\rho A_3} = \frac{\dot{m}_3}{\rho(\pi D_3^2/4)} = \frac{8\ \text{kg/s}}{(1000\ \text{kg/m}^3)[\pi(0.03\ \text{m})^2/4]} = 11.3\ \text{m/s}$$

We take the entire U-section as the control volume. We designate the horizontal coordinate by *x* with the direction of incoming flow as being the positive direction and the vertical coordinate by *z*. The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta\dot{m}\vec{V} - \sum_{\text{in}} \beta\dot{m}\vec{V}$. We let the *x*- and *z*- components of the anchoring force of the cone be $F_{Rx}$ and $F_{Rz}$, and assume them to be in the positive directions. Then the momentum equations along the *x* and *z* axes become

$$F_{Rx} + P_1A_1 + P_2A_2 = \beta\dot{m}_2(-V_2) - \beta\dot{m}_1V_1 \quad \rightarrow \quad F_{Rx} = -P_1A_1 - P_2A_2 - \beta(\dot{m}_2V_2 + \dot{m}_1V_1)$$
$$F_{Rz} + 0 = \dot{m}_3V_3 - 0 \quad \rightarrow \quad F_{Rz} = \beta\dot{m}_3V_3$$

Substituting the given values,

$$F_{Rx} = -[(200-100)\ \text{kN/m}^2]\frac{\pi(0.05\ \text{m})^2}{4} - [(150-100)\ \text{kN/m}^2]\frac{\pi(0.10\ \text{m})^2}{4}$$
$$-1.03\left[(22\ \text{kg/s})(2.80\ \text{m/s})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) + (30\ \text{kg/s})(15.3\ \text{m/s})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)\right]$$
$$= -0.733\ \text{kN} = \mathbf{-733\ N}$$

$$F_{Rz} = 1.03(8\ \text{kg/s})(11.3\ \text{m/s})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = \mathbf{93.1 N}$$

The negative value for $F_{Rx}$ indicates the assumed direction is wrong, and should be reversed. Therefore, a force of 733 N acts on the flanges in the opposite direction. A vertical force of 93.1 N acts on the flange in the vertical direction.

***Discussion*** To assess the significance of gravity forces, we estimate the weight of the weight of water in the U-turn and compare it to the vertical force. Assuming the length of the U-turn to be 0.5 m and the average diameter to be 7.5 cm, the mass of the water becomes

$$m = \rho V = \rho AL = \rho\frac{\pi D^2}{4}L = (1000\ \text{kg/m}^3)\frac{\pi(0.075\ \text{m})^2}{4}(0.5\ \text{m}) = 2.2\ \text{kg}$$

whose weight is 2.2×9.81 = 22 N, which is much less than 93.1, but still significant. Therefore, disregarding the gravitational effects is a reasonable assumption if great accuracy is not required.

**6-59**

**Solution** A fireman was hit by a nozzle held by a tripod with a rated holding force. The accident is to be investigated by calculating the water velocity, the flow rate, and the nozzle velocity.

***Assumptions*** **1** The flow is steady and incompressible. **2** The water jet is exposed to the atmosphere, and thus the pressure of the water jet is the atmospheric pressure, which is disregarded since it acts on all surfaces. **3** Gravitational effects and vertical forces are disregarded since the horizontal resistance force is to be determined. **4** Jet flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

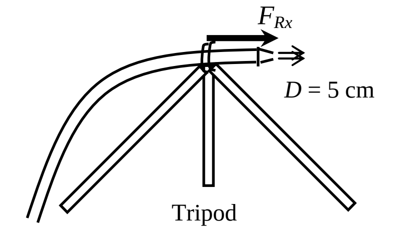
***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** We take the nozzle and the horizontal portion of the hose as the system such that water enters the control volume vertically and outlets horizontally (this way the pressure force and the momentum flux at the inlet are in the vertical direction, with no contribution to the force balance in the horizontal direction, and designate the entrance by 1 and the outlet by 2. We also designate the horizontal coordinate by *x* (with the direction of flow as being the positive direction).

The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{out} \beta \dot{m}\vec{V} - \sum_{in} \beta \dot{m}\vec{V}$. We let the horizontal force applied by the tripod to the nozzle to hold it be $F_{Rx}$, and assume it to be in the positive *x* direction. Then the momentum equation along the *x* direction becomes

$$F_{Rx} = \dot{m}V_e - 0 = \dot{m}V = \rho AVV = \rho \frac{\pi D^2}{4} V^2 \quad \rightarrow \quad (1800\text{ N})\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right) = (1000\text{ kg/m}^3)\frac{\pi(0.05\text{ m})^2}{4} V^2$$

Solving for the water outlet velocity gives $V$ = **30.3 m/s.** Then the water flow rate becomes




$$\dot{V} = AV = \frac{\pi D^2}{4} V = \frac{\pi (0.05\text{ m})^2}{4}(30.3\text{ m/s}) = \mathbf{0.0595\ m^3/s}$$

When the nozzle was released, its acceleration must have been

$$a_{\text{nozzle}} = \frac{F}{m_{\text{nozzle}}} = \frac{1800\text{ N}}{10\text{ kg}}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right) = 180\text{ m/s}^2$$

Assuming the reaction force acting on the nozzle and thus its acceleration to remain constant, the time it takes for the nozzle to travel 60 cm and the nozzle velocity at that moment were (note that both the distance *x* and the velocity *V* are zero at time *t* = 0)

$$x = \tfrac{1}{2}at^2 \quad \rightarrow \quad t = \sqrt{\frac{2x}{a}} = \sqrt{\frac{2(0.6\text{ m})}{180\text{ m/s}^2}} = 0.0816\ s$$

$$V = at = (180\text{ m/s}^2)(0.0816\text{ s}) = \mathbf{14.7\ m/s}$$

Thus we conclude that the nozzle hit the fireman with a velocity of 14.7 m/s.

***Discussion*** Engineering analyses such as this one are frequently used in accident reconstruction cases, and they often form the basis for judgment in courts.

**6-60**

**Solution** During landing of an airplane, the thrust reverser is lowered in the path of the exhaust jet, which deflects the exhaust and provides braking. The thrust of the engine and the braking force produced after the thrust reverser is deployed are to be determined.

***Assumptions*** **1** The flow of exhaust gases is steady and one-dimensional. **2** The exhaust gas stream is exposed to the atmosphere, and thus its pressure is the atmospheric pressure. **3** The velocity of exhaust gases remains constant during reversing. **4** Jet flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Analysis*** (*a*) The thrust exerted on an airplane is simply the momentum flux of the combustion gases in the reverse direction,

$$\text{Thrust} = \dot{m}_{ex}V_{ex} = (18\text{ kg/s})(250\text{ m/s})\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = \mathbf{4500\ N}$$

(*b*) We take the thrust reverser as the control volume such that it cuts through both exhaust streams normally and the connecting bars to the airplane, and the direction of airplane as the positive direction of *x* axis. The momentum equation for steady one-dimensional flow in the *x* direction reduces to

$$\sum \vec{F} = \sum_{\text{out}} \beta\dot{m}\vec{V} - \sum_{\text{in}} \beta\dot{m}\vec{V} \quad \rightarrow \quad F_{Rx} = \dot{m}(V)\cos 20° - \dot{m}(-V) \quad \rightarrow \quad F_{Rx} = (1+\cos 20°)\dot{m}V_i$$

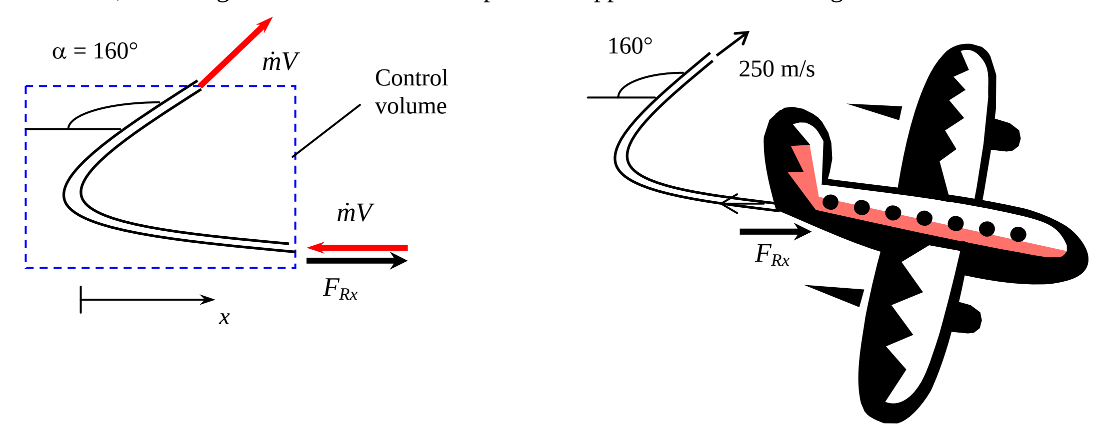
Substituting, the reaction force is determined to be

$$F_{Rx} = (1+\cos 20°)(18\text{ kg/s})(250\text{ m/s}) = 8729\text{ N}$$

The breaking force acting on the plane is equal and opposite to this force,

$$F_{\text{breaking}} = 8729\text{ N} \cong \mathbf{8730\ \ N}$$

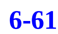
Therefore, a braking force of 8730 N develops in the opposite direction to flight.




***Discussion*** This problem can be solved more generally by measuring the reversing angle from the direction of exhaust gases (α = 0 when there is no reversing). When α < 90°, the reversed gases are discharged in the negative *x* direction, and the momentum equation reduces to

$$F_{Rx} = \dot{m}(-V)\cos\alpha - \dot{m}(-V) \quad \rightarrow \quad F_{Rx} = (1-\cos\alpha)\dot{m}V_i$$

This equation is also valid for α >90° since cos(180°-α) = - cosα. Using α = 160°, for example, gives $F_{Rx} = (1-\cos 160)\dot{m}V_i = (1+\cos 20)\dot{m}V_i$ , which is identical to the solution above.



**6-61** 

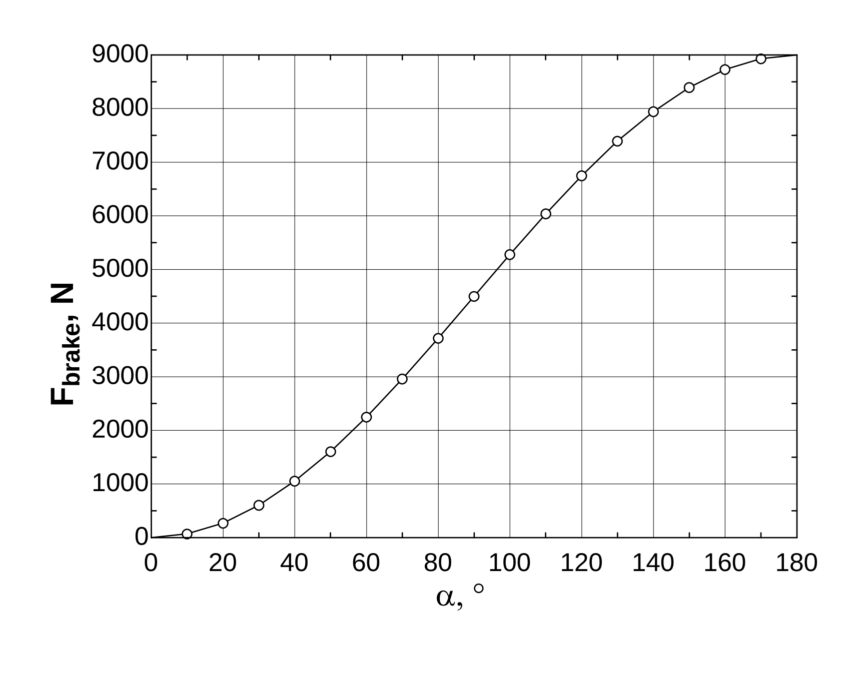
**Solution** The previous problem is reconsidered. The effect of thrust reverser angle on the braking force exerted on the airplane as the reverser angle varies from 0 (no reversing) to 180° (full reversing) in increments of 10° is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
V_jet=250 "m/s"

m_dot=18 "kg/s"
F_Rx=(1-cos(alpha))*m_dot*V_jet "N"
```

| Reversing angle, $\alpha$° | Braking force $F_{\text{brake}}$, N |
|---|---|
| 0 | 0 |
| 10 | 68 |
| 20 | 271 |
| 30 | 603 |
| 40 | 1053 |
| 50 | 1607 |
| 60 | 2250 |
| 70 | 2961 |
| 80 | 3719 |
| 90 | 4500 |
| 100 | 5281 |
| 110 | 6039 |
| 120 | 6750 |
| 130 | 7393 |
| 140 | 7947 |
| 150 | 8397 |
| 160 | 8729 |
| 170 | 8932 |
| 180 | 9000 |



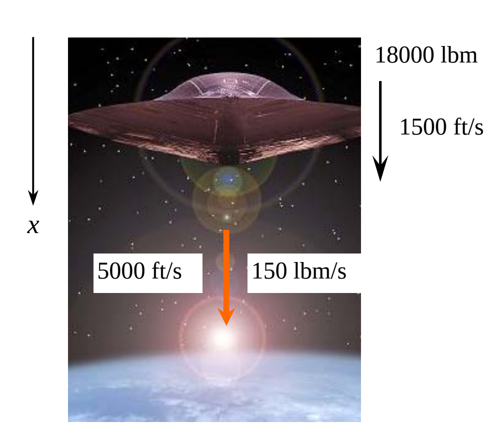

***Discussion*** As expected, the braking force is zero when the angle is zero (no deflection), and maximum when the angle is 180° (completely reversed). Of course, it is impossible to completely reverse the flow, since the jet exhaust cannot be directed back into the engine.

**6-62E**

**Solution** The rocket of a spacecraft is fired in the opposite direction to motion. The deceleration, the velocity change, and the thrust are to be determined.

***Assumptions*** **1** The flow of combustion gases is steady and one-dimensional during the firing period, but the flight of the spacecraft is unsteady. **2** There are no external forces acting on the spacecraft, and the effect of pressure force at the nozzle outlet is negligible. **3** The mass of discharged fuel is negligible relative to the mass of the spacecraft, and thus the spacecraft may be treated as a solid body with a constant mass. **4** The nozzle is well-designed such that the effect of the momentum-flux correction factor is negligible, and thus $\beta \cong 1$.

***Analysis*** (*a*) We choose a reference frame in which the control volume moves with the spacecraft. Then the velocities of fluid steams become simply their relative velocities (relative to the moving body). We take the direction of motion of the spacecraft as the positive direction along the *x* axis. There are no external forces acting on the spacecraft, and its mass is nearly constant. Therefore, the spacecraft can be treated as a solid body with constant mass, and the momentum equation in this case is




$$0 = \frac{d(m\vec{V})_{CV}}{dt} + \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V} \quad \rightarrow \quad m_{\text{space}} \frac{d\vec{V}_{\text{space}}}{dt} = -\dot{m}_f \vec{V}_f$$

Noting that the motion is on a straight line and the discharged gases move in the positive *x* direction (to slow down the spacecraft), we write the momentum equation using magnitudes as

$$m_{\text{space}} \frac{dV_{\text{space}}}{dt} = -\dot{m}_f V_f \quad \rightarrow \quad \frac{dV_{\text{space}}}{dt} = -\frac{\dot{m}_f}{m_{\text{space}}} V_f$$

Substituting, the deceleration of the spacecraft during the first 5 seconds is determined to be

$$a_{\text{space}} = \frac{dV_{\text{space}}}{dt} = -\frac{\dot{m}_f}{m_{\text{space}}} V_f = -\frac{150 \text{ lbm/s}}{18{,}000 \text{ lbm}} (5000 \text{ ft/s}) = \mathbf{-41.7 \text{ ft/s}^2}$$

(*b*) Knowing the deceleration, which is constant, the velocity change of the spacecraft during the first 5 seconds is determined from the definition of acceleration $a_{\text{space}} = dV_{\text{space}} / dt$ to be

$$dV_{\text{space}} = a_{\text{space}} dt \quad \rightarrow \quad \Delta V_{\text{space}} = a_{\text{space}} \Delta t = (-41.7 \text{ ft/s}^2)(5 \text{ s}) = \mathbf{-209 \text{ ft/s}}$$

(*c*) The thrust exerted on the system is simply the momentum flux of the combustion gases in the reverse direction,

$$\text{Thrust} = F_R = -\dot{m}_f V_f = -(150 \text{ lbm/s})(5000 \text{ ft/s})\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm} \cdot \text{ft/s}^2}\right) = -23{,}290 \text{ lbf} \cong \mathbf{-23{,}300 \text{ lbf}}$$

Therefore, if this spacecraft were attached somewhere, it would exert a force of 23,300 lbf (equivalent to the weight of 23,300 lbm of mass on earth) to its support in the negative *x* direction.

***Discussion*** In Part (*b*) we approximate the deceleration as constant. However, since mass is lost from the spacecraft during the time in which the jet is on, a more accurate solution would involve solving a differential equation. Here, the time span is short, and the lost mass is likely negligible compared to the total mass of the spacecraft, so the more complicated analysis is not necessary.

**6-63**

**Solution** A horizontal water jet strikes a vertical stationary flat plate normally at a specified velocity. For a given flow velocity, the anchoring force needed to hold the plate in place is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The water splatters off the sides of the plate in a plane normal to the jet. **3** The water jet is exposed to the atmosphere, and thus the pressure of the water jet and the splattered water is the atmospheric pressure which is disregarded since it acts on the entire control surface. **4** The vertical forces and momentum fluxes are not considered since they have no effect on the horizontal reaction force. **5** Jet flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

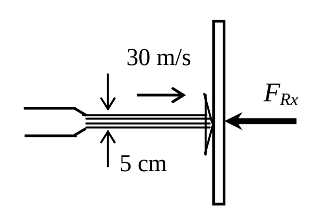
***Analysis*** We take the plate as the control volume such that it contains the entire plate and cuts through the water jet and the support bar normally, and the direction of flow as the positive direction of $x$ axis. We take the reaction force to be in the negative $x$ direction. The momentum equation for steady one-dimensional flow in the $x$ (flow) direction reduces in this case to



$$\sum \vec{F} = \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V} \quad \rightarrow \quad -F_{Rx} = -\dot{m}_i V_i \quad \rightarrow \quad F_{Rx} = \dot{m}V$$

We note that the reaction force acts in the opposite direction to flow, and we should not forget the negative sign for forces and velocities in the negative $x$-direction. The mass flow rate of water is

$$\dot{m} = \rho \dot{V} = \rho AV = \rho \frac{\pi D^2}{4} V = (1000\ \text{kg/m}^3)\frac{\pi (0.05\ \text{m})^2}{4}(30\ \text{m/s}) = 58.90\ \text{kg/s}$$




Substituting, the reaction force is determined to be

$$F_{Rx} = (58.90\ \text{kg/s})(30\ \text{m/s}) = 1767\ \text{N} \cong \mathbf{1770\ N}$$

Therefore, a force of 1770 N must be applied to the plate in the opposite direction to the flow to hold it in place.

***Discussion*** In reality, some water may be scattered back, and this would add to the reaction force of water.

**6-64**

**Solution** A water jet hits a stationary cone, such that the flow is diverted equally in all directions at 45°. The force required to hold the cone in place against the water stream is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The water jet is exposed to the atmosphere, and thus the pressure of the water jet before and after the split is the atmospheric pressure which is disregarded since it acts on all surfaces. **3** The gravitational effects are disregarded. **4** Jet flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** The mass flow rate of water jet is

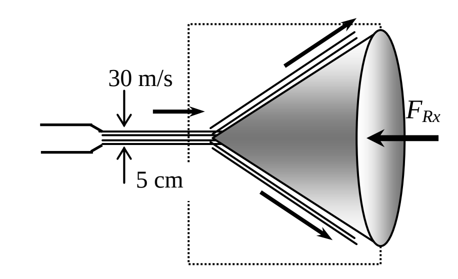
$$\dot{m} = \rho \dot{V} = \rho A V = \rho \frac{\pi D^2}{4} V = (1000\ \text{kg/m}^3) \frac{\pi (0.05\ \text{m})^2}{4} (30\ \text{m/s}) = 58.90\ \text{kg/s}$$

We take the diverting section of water jet, including the cone as the control volume, and designate the entrance by 1 and the outlet after divergence by 2. We also designate the horizontal coordinate by $x$ with the direction of flow as being the positive direction and the vertical coordinate by $y$.

The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta \dot{m} \vec{V} - \sum_{\text{in}} \beta \dot{m} \vec{V}$. We let the $x$- and $y$- components of the anchoring force of the cone be $F_{Rx}$ and $F_{Ry}$, and assume them to be in the positive directions. Noting that $V_2 = V_1 = V$ and $\dot{m}_2 = \dot{m}_1 = \dot{m}$, the momentum equations along the $x$ and $y$ axes become

$$F_{Rx} = \dot{m} V_2 \cos\theta - \dot{m} V_1 = \dot{m} V (\cos\theta - 1)$$

$$F_{Ry} = 0 \quad \text{(because of symmetry about x axis)}$$




Substituting the given values,

$$F_{Rx} = (58.90\ \text{kg/s})(30\ \text{m/s})(\cos 45° - 1)\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)$$

$$= \mathbf{-518\ N}$$

$$F_{Ry} = \mathbf{0}$$

The negative value for $F_{Rx}$ indicates that the assumed direction is wrong, and should be reversed. Therefore, a force of 518 N must be applied to the cone in the opposite direction to flow to hold it in place. No holding force is necessary in the vertical direction due to symmetry and neglecting gravitational effects.

***Discussion*** In reality, the gravitational effects will cause the upper part of flow to slow down and the lower part to speed up after the split. But for short distances, these effects are negligible.

**6-65**

**Solution** An ice skater is holding a flexible hose (essentially weightless) which directs a stream of water horizontally at a specified velocity. The velocity and the distance traveled in 5 seconds, and the time it takes to move 5 m and the velocity at that moment are to be determined.

***Assumptions*** **1** Friction between the skates and ice is negligible. **2** The flow of water is steady and one-dimensional (but the motion of skater is unsteady). **3** The ice skating arena is level, and the water jet is discharged horizontally. **4** The mass of the hose and the water in it is negligible. **5** The skater is standing still initially at $t = 0$. **6** Jet flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** (*a*) The mass flow rate of water through the hose is

$$\dot{m} = \rho A V = \rho \frac{\pi D^2}{4} V = (1000\ \text{kg/m}^3)\frac{\pi (0.02\ \text{m})^2}{4}(10\ \text{m/s}) = 3.14\ \text{kg/s}$$

The thrust exerted on the skater by the water stream is simply the momentum flux of the water stream, and it acts in the reverse direction,

$$F = \text{Thrust} = \dot{m}V = (3.14\ \text{kg/s})(10\ \text{m/s})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = 31.4\ \text{N}\ \ (\text{constant})$$

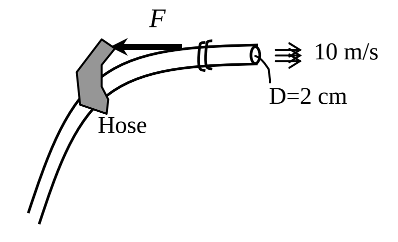
The acceleration of the skater is determined from Newton’s 2$^{\text{nd}}$ law of motion $F = ma$ where $m$ is the mass of the skater,

$$a = \frac{F}{m} = \frac{31.4\ \text{N}}{60\ \text{kg}}\left(\frac{1\ \text{kg}\cdot\text{m/s}^2}{1\ \text{N}}\right) = 0.523\ \text{m/s}^2$$

Note that thrust and thus the acceleration of the skater is constant. The velocity of the skater and the distance traveled in 5 s are

$$V_{\text{skater}} = at = (0.523\ \text{m/s}^2)(5\ \text{s}) = \mathbf{2.62\ m/s}$$

$$x = \tfrac{1}{2}at^2 = \tfrac{1}{2}(0.523\ \text{m/s}^2)(5\ \text{s})^2 = \mathbf{6.54\ m}$$




(*b*) The time it will take to move 5 m and the velocity at that moment are

$$x = \tfrac{1}{2}at^2 \quad \rightarrow \quad t = \sqrt{\frac{2x}{a}} = \sqrt{\frac{2(5\ \text{m})}{0.523\ \text{m/s}^2}} = \mathbf{4.4\ s}$$

$$V_{\text{skater}} = at = (0.523\ \text{m/s}^2)(4.4\ \text{s}) = \mathbf{2.3\ m/s}$$

***Discussion*** In reality, the velocity of the skater will be lower because of friction on ice and the resistance of the hose to follow the skater. Also, in the $\beta \dot{m} V$ expressions, $V$ is the fluid stream speed relative to a fixed point. Therefore, the correct expression for thrust is $F = \dot{m}(V_{jet} - V_{skater})$, and the analysis above is valid only when the skater speed is low relative to the jet speed. An exact analysis would result in a differential equation.

**6-66**

**Solution** Indiana Jones is to ascend a building by building a platform, and mounting four water nozzles pointing down at each corner. The minimum water jet velocity needed to raise the system, the time it will take to rise to the top of the building and the velocity of the system at that moment, the additional rise when the water is shut off, and the time he has to jump from the platform to the roof are to be determined.

***Assumptions*** **1** The air resistance is negligible. **2** The flow of water is steady and one-dimensional (but the motion of platform is unsteady). **3** The platform is still initially at $t$ = 0. **4** Jet flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Properties*** We take the density of water to be 1000 kg/m$^3$.

***Analysis*** ($a$) The total mass flow rate of water through the 4 hoses and the total weight of the platform are

$$\dot{m} = \rho AV = 4\rho \frac{\pi D^2}{4} V = 4(1000\ \text{kg/m}^3)\frac{\pi(0.05\ \text{m})^2}{4}(15\ \text{m/s}) = 118\ \text{kg/s}$$

$$W = mg = (150\ \text{kg})(9.81\ \text{m/s}^2)\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = 1472\ \text{N}$$

We take the platform as the system. The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta\dot{m}\vec{V} - \sum_{\text{in}} \beta\dot{m}\vec{V}$ . The minimum water jet velocity needed to raise the platform is determined by setting the net force acting on the platform equal to zero,

$$-W = \dot{m}(-V_{\min}) - 0 \quad \rightarrow \quad W = \dot{m}V_{\min} = \rho AV_{\min}V_{\min} = 4\rho\frac{\pi D^2}{4}V_{\min}^2$$

Solving for $V_{\min}$ and substituting,

$$V_{\min} = \sqrt{\frac{W}{\rho\pi D^2}} = \sqrt{\frac{1472\ \text{N}}{(1000\ \text{kg/m}^3)\pi(0.05\ \text{m})^2}\left(\frac{1\text{kg}\cdot\text{m/s}^2}{1\text{N}}\right)} = \mathbf{13.7\ m/s}$$

($b$) We let the vertical reaction force (assumed upwards) acting on the platform be $F_{Rz}$. Then the momentum equation in the vertical direction becomes

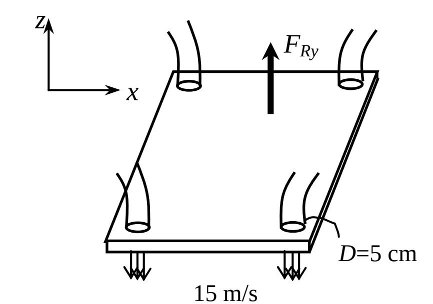
$$F_{Rz} - W = \dot{m}(-V) - 0 = \dot{m}V \quad \rightarrow \quad F_{Rz} = W - \dot{m}V = (1472\ \text{N}) - (118\ \text{kg/s})(15\ \text{m/s})\left(\frac{1\text{kg}\cdot\text{m/s}^2}{1\text{N}}\right) = -298\ \text{N}$$

The upward thrust acting on the platform is equal and opposite to this reaction force, and thus $F$ = 298 N. Then the acceleration and the ascending time to rise 10 m and the velocity at that moment become

$$a = \frac{F}{m} = \frac{298\ \text{N}}{150\ \text{kg}}\left(\frac{1\ \text{kg}\cdot\text{m/s}^2}{1\ \text{N}}\right) = 2.0\ \text{m/s}^2$$

$$x = \tfrac{1}{2}at^2 \quad \rightarrow \quad t = \sqrt{\frac{2x}{a}} = \sqrt{\frac{2(10\ \text{m})}{2\ \text{m/s}^2}} = \mathbf{3.2\ s}$$

$$V = at = (2\ \text{m/s}^2)(3.2\ \text{s}) = \mathbf{6.4\ m/s}$$




($c$) When water is shut off at 10 m height (where the velocity is 6.4 m/s), the platform will decelerate under the influence of gravity, and the time it takes to come to a stop and the additional rise above 10 m become

$$V = V_0 - gt = 0 \quad \rightarrow \quad t = \frac{V_0}{g} = \frac{6.4\ \text{m/s}}{9.81\ \text{m/s}^2} = \mathbf{0.65\ s}$$

$$z = V_0 t - \tfrac{1}{2}gt^2 = (6.4\ \text{m/s})(0.65\ \text{s}) - \tfrac{1}{2}(9.81\ \text{m/s}^2)(0.65\ \text{s})^2 = \mathbf{2.1\,m}$$

Therefore, Jones has 2×0.65 = 1.3 s to jump off from the platform to the roof since it takes another 0.65 s for the platform to descend to the 10 m level.

***Discussion*** Like most stunts in the Indiana Jones movies, this would not be practical in reality.

**6-67E**

**Solution** A box-enclosed fan is faced down so the air blast is directed downwards, and it is to be hovered by increasing the blade rpm. The required blade rpm, air outlet velocity, the volumetric flow rate, and the minimum mechanical power are to be determined.

***Assumptions*** **1** The flow of air is steady and incompressible. **2** The air leaves the blades at a uniform velocity at atmospheric pressure, and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$. **3** Air approaches the blades from the top through a large area at atmospheric pressure with negligible velocity. **4** The frictional effects are negligible, and thus the entire mechanical power input is converted to kinetic energy of air (no conversion to thermal energy through frictional effects). **5** The change in air pressure with elevation is negligible because of the low density of air. **6** There is no acceleration of the fan, and thus the lift generated is equal to the total weight.

***Properties*** The density of air is given to be 0.078 lbm/ft$^3$.

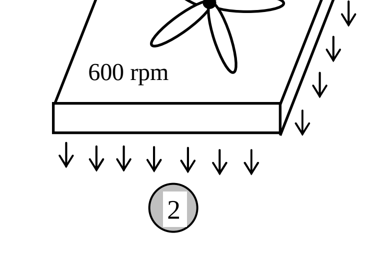
***Analysis*** (*a*) We take the control volume to be a vertical hyperbolic cylinder bounded by streamlines on the sides with air entering through the large cross-section (section 1) at the top and the fan located at the narrow cross-section at the bottom (section 2), and let its centerline be the *z* axis with upwards being the positive direction.

The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{out} \beta \dot{m} \vec{V} - \sum_{in} \beta \dot{m} \vec{V}$. Noting that the only force acting on the control volume is the total weight *W* and it acts in the negative *z* direction, the momentum equation along the *z* axis gives

$$-W = \dot{m}(-V_2) - 0 \quad \rightarrow \quad W = \dot{m}V_2 = (\rho A V_2)V_2 = \rho A V_2^2 \quad \rightarrow \quad V_2 = \sqrt{\frac{W}{\rho A}}$$

where *A* is the blade span area,

$$A = \pi D^2/4 = \pi (3\text{ ft})^2/4 = 7.069\text{ ft}^2$$




Then the discharge velocity to produce 5 lbf of upward force becomes

$$V_2 = \sqrt{\frac{5\text{ lbf}}{(0.078\text{ lbm/ft}^3)(7.069\text{ ft}^2)}\left(\frac{32.2\text{ lbm}\cdot\text{ft/s}^2}{1\text{ lbf}}\right)} = \mathbf{17.1\ ft/s}$$

(*b*) The volume flow rate and the mass flow rate of air are determined from their definitions,

$$\dot{V} = AV_2 = (7.069\text{ ft}^2)(17.1\text{ ft/s}) = \mathbf{121\ ft^3/s}$$

$$\dot{m} = \rho \dot{V} = (0.078\text{ lbm/ft}^3)(121\text{ ft}^3/\text{s}) = 9.43\text{ lbm/s}$$

(*c*) Noting that $P_1 = P_2 = P_{atm}$, $V_1 \cong 0$, the elevation effects are negligible, and the frictional effects are disregarded, the energy equation for the selected control volume reduces to

$$\dot{m}\left(\frac{P_1}{\rho} + \frac{V_1^2}{2} + gz_1\right) + \dot{W}_{\text{pump, u}} = \dot{m}\left(\frac{P_2}{\rho} + \frac{V_2^2}{2} + gz_2\right) + \dot{W}_{\text{turbine}} + \dot{E}_{\text{mech,loss}} \quad \rightarrow \quad \dot{W}_{\text{fan, u}} = \dot{m}\frac{V_2^2}{2}$$

Substituting,

$$\dot{W}_{\text{fan,u}} = \dot{m}\frac{V_2^2}{2} = (9.43\text{ lbm/s})\frac{(18.0\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1\text{ W}}{0.73756\text{ lbf}\cdot\text{ft/s}}\right) = \mathbf{64.3\ W}$$

Therefore, the minimum mechanical power that must be supplied to the air stream is 64.3 W.

***Discussion*** The actual power input to the fan will be considerably larger than the calculated power input because of the fan inefficiency in converting mechanical work to kinetic energy.

**6-68**

**Solution** A parachute slows a soldier from his terminal velocity $V_T$ to his landing velocity of $V_F$. A relation is to be developed for the soldier's velocity after he opens the parachute at time $t = 0$.

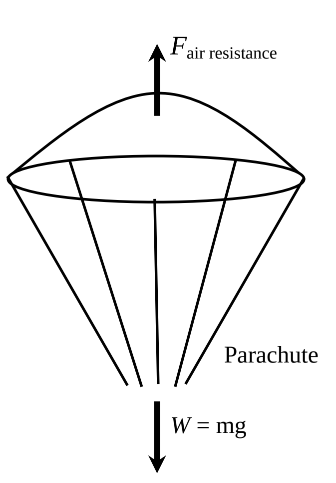
***Assumptions*** **1** The air resistance is proportional to the velocity squared (i.e. $F = -kV^2$). **2** The variation of the air properties with altitude is negligible. **3** The buoyancy force applied by air to the person (and the parachute) is negligible because of the small volume occupied and the low density of air. **4** The final velocity of the soldier is equal to its terminal velocity with his parachute open.

***Analysis*** The terminal velocity of a free falling object is reached when the air resistance (or air drag) equals the weight of the object, less the buoyancy force applied by the fluid, which is negligible in this case,

$$F_{\text{air resistance}} = W \quad \rightarrow \quad kV_F^2 = mg \quad \rightarrow \quad k = \frac{mg}{V_F^2}$$

This is the desired relation for the constant of proportionality $k$. When the parachute is deployed and the soldier starts to decelerate, the net downward force acting on him is his weight less the air resistance,

$$F_{\text{net}} = W - F_{\text{air resistance}} = mg - kV^2 = mg - \frac{mg}{V_F^2}V^2 = mg\left(1 - \frac{V^2}{V_F^2}\right)$$




Substituting it into Newton's 2nd law relation $F_{\text{net}} = ma = m\frac{dV}{dt}$ gives

$$mg\left(1 - \frac{V^2}{V_F^2}\right) = m\frac{dV}{dt}$$

Canceling $m$ and separating variables, and integrating from $t = 0$ when $V = V_T$ to $t = t$ when $V = V$ gives

$$\frac{dV}{1 - V^2/V_F^2} = g\,dt \quad \rightarrow \quad \int_{V_T}^{V} \frac{dV}{V_F^2 - V^2} = \frac{g}{V_F^2}\int_0^t dt$$

Using $\int \frac{dx}{a^2 - x^2} = \frac{1}{2a}\ln\frac{a+x}{a-x}$ from integral tables and applying the integration limits,

$$\frac{1}{2V_F}\left(\ln\frac{V_F + V}{V_F - V} - \ln\frac{V_F + V_T}{V_F - V_T}\right) = \frac{gt}{V_F^2}$$

Rearranging, the velocity can be expressed explicitly as a function of time as

$$\boxed{V = V_F\,\frac{V_T + V_F + (V_T - V_F)e^{-2gt/V_F}}{V_T + V_F - (V_T - V_F)e^{-2gt/V_F}}}$$

***Discussion*** Note that as $t \rightarrow \infty$, the velocity approaches the landing velocity of $V_F$, as expected.

**6-69**

**Solution** An empty cart is to be driven by a horizontal water jet that enters from a hole at the rear of the cart. A relation is to be developed for cart velocity versus time.

***Assumptions*** **1** The flow of water is steady, one-dimensional, incompressible, and horizontal. **2** All the water which enters the cart is retained. **3** The path of the cart is level and frictionless. **4** The cart is initially empty and stationary, and thus $V = 0$ at time $t = 0$. **5** Friction between water jet and air is negligible, and the entire momentum of water jet is used to drive the cart with no losses. **6** Jet flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Analysis*** We note that the water jet velocity $V_J$ is constant, but the car velocity $V$ is variable. Noting that $\dot{m} = \rho A(V_J - V)$ where $A$ is the cross-sectional area of the water jet and $V_J$ - $V$ is the velocity of the water jet relative to the cart, the mass of water in the cart at any time $t$ is

$$m_w = \int_0^t \dot{m}dt = \int_0^t \rho A(V_J - V)dt = \rho A V_J t - \rho A\int_0^t V dt \quad (1)$$

Cart
$m_0$
Waterjet
$V$

Also,

$$\frac{dm_w}{dt} = \dot{m} = \rho A(V_J - V)$$

We take the cart as the moving control volume. The net force acting on the cart in this case is equal to the momentum flux of the water jet. Newton's 2nd law $F = ma = d(mV)/dt$ in this case can be expressed as

$$F = \frac{d(m_{\text{total}}V)}{dt} \quad \text{where} \quad F = \sum_{\text{in}} \beta\dot{m}V - \sum_{\text{out}} \beta\dot{m}V = (\dot{m}V)_{in} = \dot{m}V_J = \rho A(V_J - V)V_J$$

and

$$\frac{d(m_{\text{total}}V)}{dt} = \frac{d[(m_c + m_w)V]}{dt} = m_c\frac{dV}{dt} + \frac{d(m_w V)}{dt} = m_c\frac{dV}{dt} + m_w\frac{dV}{dt} + V\frac{dm_w}{dt}$$

$$= (m_c + m_w)\frac{dV}{dt} + \rho A(V_J - V)V$$

Note that in $\beta\dot{m}V$ expressions, we used the fluid stream velocity relative to a fixed point. Substituting,

$$\rho A(V_J - V)V_J = (m_c + m_w)\frac{dV}{dt} + \rho A(V_J - V)V \quad \rightarrow \quad \rho A(V_J - V)(V_J - V) = (m_c + m_w)\frac{dV}{dt}$$

Noting that $m_w$ is a function of $t$ (as given by Eq. 1) and separating variables,

$$\frac{dV}{\rho A(V_J - V)^2} = \frac{dt}{m_c + m_w} \quad \rightarrow \quad \frac{dV}{\rho A(V_J - V)^2} = \frac{dt}{m_c + \rho A V_J t - \rho A\int_0^t V dt}$$

Integrating from $t = 0$ when $V = 0$ to $t = t$ when $V = V$ gives the desired integral,

$$\boxed{\int_0^V \frac{dV}{\rho A(V_J - V)^2} = \int_o^t \frac{dt}{m_c + \rho A V_J t - \rho A\int_0^t V dt}}$$

***Discussion*** Note that the time integral involves the integral of velocity, which complicates the solution.

**6-70**

**Solution** A plate is maintained in a horizontal position by frictionless vertical guide rails. The underside of the plate is subjected to a water jet. The minimum mass flow rate $\dot{m}_{\min}$ to just levitate the plate is to be determined, and a relation is to be obtained for the steady state upward velocity. Also, the integral that relates velocity to time when the water is first turned on is to be obtained.

***Assumptions*** **1** The flow of water is steady and one-dimensional. **2** The water jet splatters in the plane of he plate. **3** The vertical guide rails are frictionless. **4** Times are short, so the velocity of the rising jet can be considered to remain constant with height. **5** At time t = 0, the plate is at rest. **6** Jet flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

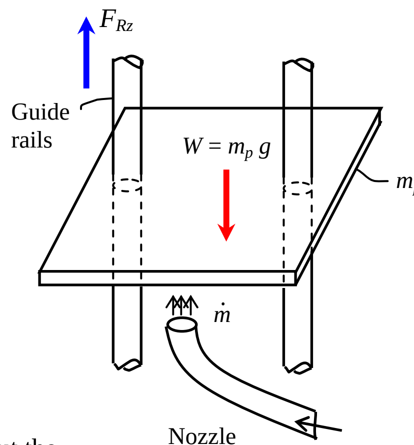
***Analysis*** (*a*) We take the plate as the system. The momentum equation for steady one-dimensional flow is $\sum \vec{F} = \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V}$. Noting that $\dot{m} = \rho A V_J$ where $A$ is the cross-sectional area of the water jet and $W = m_p g$, the minimum mass flow rate of water needed to raise the plate is determined by setting the net force acting on the plate equal to zero,

$$-W = 0 - \dot{m}_{\min} V_J \quad \rightarrow \quad W = \dot{m}_{\min} V_J \;\rightarrow\; m_p g = \dot{m}_{\min}(\dot{m}_{\min} / A V_J) \rightarrow \quad \dot{m}_{\min} = \sqrt{\rho A m_p g}$$

For $\dot{m} > \dot{m}_{\min}$, a relation for the steady state upward velocity $V$ is obtained setting the upward impulse applied by water jet to the weight of the plate (during steady motion, the plate velocity $V$ is constant, and the velocity of water jet relative to plate is $V_J - V$),

$$W = \dot{m}(V_J - V) \quad \rightarrow \quad m_p g = \rho A (V_J - V)^2 \quad \rightarrow \quad V_J - V = \sqrt{\frac{m_p g}{\rho A}} \quad \rightarrow \quad V = \frac{\dot{m}}{\rho A} - \sqrt{\frac{m_p g}{\rho A}}$$

(*b*) At time $t = 0$ the plate is at rest ($V = 0$), and it is subjected to water jet with $\dot{m} > \dot{m}_{\min}$ and thus the net force acting on it is greater than the weight of the plate, and the difference between the jet impulse and the weight will accelerate the plate upwards. Therefore, Newton's 2nd law $F = ma = mdV/dt$ in this case can be expressed as




$$\dot{m}(V_J - V) - W = m_p a \quad \rightarrow \quad \rho A (V_J - V)^2 - m_p g = m_p \frac{dV}{dt}$$

Separating the variables and integrating from $t = 0$ when $V = 0$ to $t = t$ when $V = V$ gives the desired integral,

$$\int_0^V \frac{m_p dV}{\rho A (V_J - V)^2 - m_p g} = \int_{t=0}^t dt \quad \rightarrow \quad \boxed{t = \int_0^V \frac{m_p dV}{\rho A (V_J - V)^2 - m_p g}}$$

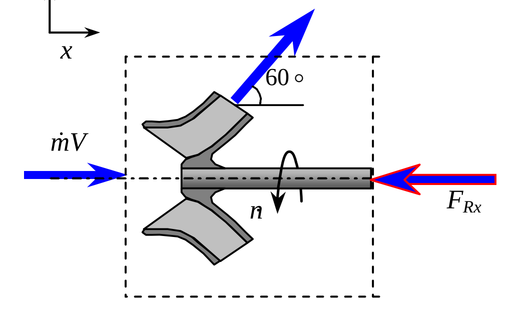
***Discussion*** This integral can be performed with the help of integral tables. But the relation obtained will be implicit in $V$.

**6-71**

**Solution** Water enters a centrifugal pump axially at a specified rate and velocity, and leaves at an angle from the axial direction. The force acting on the shaft in the axial direction is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The forces acting on the piping system in the horizontal direction are negligible. **3** The atmospheric pressure is disregarded since it acts on all surfaces. **4** Water flow is nearly uniform at the outlet and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

***Properties*** We take the density of water to be 1000 kg/m$^3$.




***Analysis*** From conservation of mass we have $\dot{m}_1 = \dot{m}_2 = \dot{m}$, and thus $\dot{V}_1 = \dot{V}_2$ and $A_{c1}V_1 = A_{c2}V_2$. Noting that the discharge area is half the inlet area, the discharge velocity is twice the inlet velocity. That is,

$$A_{c1}V_2 = \frac{A_{c1}}{A_{c2}}V_1 = 2V_1 = 2(5\text{ m/s}) = 10\text{ m/s}$$

We take the pump as the control volume, and the inlet direction of flow as the positive direction of $x$ axis. The linear momentum equation in this case in the $x$ direction reduces to

$$\sum \vec{F} = \sum_{\text{out}} \beta \dot{m}\vec{V} - \sum_{\text{in}} \beta \dot{m}\vec{V} \quad \rightarrow \quad -F_{Rx} = \dot{m}V_2 \cos\theta - \dot{m}V_1 \quad \rightarrow \quad F_{Rx} = \dot{m}(V_1 - V_2\cos\theta)$$

where the mass flow rate it

$$\dot{m} = \rho\dot{V} = (1000\text{ kg/m}^3)(0.20\text{ m}^3\text{/s}) = 200\text{ kg/s}$$

Substituting the known quantities, the reaction force is determined to be (note that cos60° = 0.5)

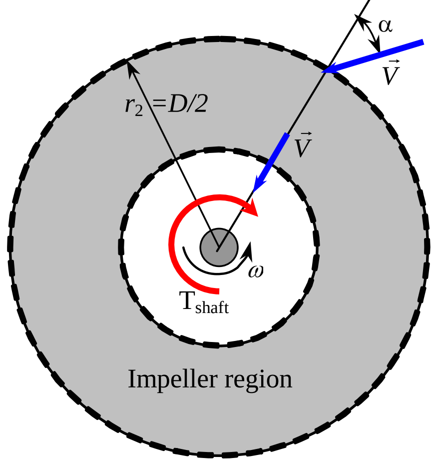
$$F_{\text{Rx}} = (200\text{ kg/s})[(5\text{ m/s}) - (10\text{ m/s})\cos 60]\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = \mathbf{0}$$

***Discussion*** Note that at this angle of discharge, the bearing is not subjected to any horizontal loading. Therefore, the loading in the system can be controlled by adjusting the discharge angle.

---

**6-72**

**Solution** Water enters the impeller of a turbine through its outer edge of diameter $D$ with velocity $V$ making an angle $\alpha$ with the radial direction at a mass flow rate of $\dot{m}$, and leaves the impeller in the radial direction. The maximum power that can be generated is to be shown to be $\dot{W}_{\text{shaft}} = \pi\dot{n}\dot{m}DV\sin\alpha$.




***Assumptions*** **1** The flow is steady in the mean. **2** Irreversible losses are negligible.

***Analysis*** We take the impeller region as the control volume. The tangential velocity components at the inlet and the outlet are $V_{1,t} = 0$ and $V_{2,t} = V\sin\alpha$.

Normal velocity components as well pressure acting on the inner and outer circumferential areas pass through the shaft center, and thus they do not contribute to torque. Only the tangential velocity components contribute to torque, and the application of the angular momentum equation gives

$$T_{\text{shaft}} = \dot{m}(r_2V_{2,t} - r_1V_{1,t}) = \dot{m}r_2V_{2,t} - 0 = \dot{m}D(V\sin\alpha)/2$$

The angular velocity of the propeller is $\omega = 2\pi\dot{n}$. Then the shaft power becomes

$$\dot{W}_{\text{shaft}} = \omega T_{\text{shaft}} = 2\pi\dot{n}\dot{m}D(V\sin\alpha)/2$$

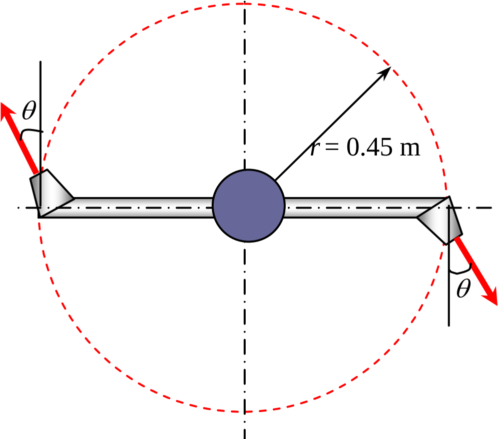
Simplifying, the maximum power generated becomes $\boxed{\dot{W}_{\text{shaft}} = \pi\dot{n}\dot{m}DV\sin\alpha}$ which is the desired relation.

***Discussion*** The actual power is less than this due to irreversible losses that are not taken into account in our analysis.

---

**6-73**

**Solution** A two-armed sprinkler is used to water a garden. For specified flow rate and discharge angles, the rates of rotation of the sprinkler head are to be determined.




***Assumptions*** **1** The flow is uniform and cyclically steady (i.e., steady from a frame of reference rotating with the sprinkler head). **2** The water is discharged to the atmosphere, and thus the gage pressure at the nozzle outlet is zero. **3** Frictional effects and air drag of rotating components are neglected. **4** The nozzle diameter is small compared to the moment arm, and thus we use average values of radius and velocity at the outlet.

***Properties*** We take the density of water to be 1000 kg/m$^3$ = 1 kg/L.

***Analysis*** We take the disk that encloses the sprinkler arms as the control volume, which is a stationary control volume. The conservation of mass equation for this steady flow system is $\dot{m}_1 = \dot{m}_2 = \dot{m}$. Noting that the two nozzles are identical, we have $\dot{m}_{\text{nozzle}} = \dot{m}/2$ or $\dot{V}_{\text{nozzle}} = \dot{V}_{\text{total}}/2$ since the density of water is constant. The average jet outlet velocity relative to the nozzle is

$$V_{\text{jet}} = \frac{\dot{V}_{\text{nozzle}}}{A_{\text{jet}}} = \frac{60\text{ L/s}}{2[\pi(0.02\text{ m})^2/4]}\left(\frac{1\text{ m}^3}{1000\text{ L}}\right) = 95.49\text{ m/s}$$

The angular momentum equation can be expressed as $\sum M = \sum_{\text{out}} r\dot{m}V - \sum_{\text{in}} r\dot{m}V$. Noting that there are no external moments acting, the angular momentum equation about the axis of rotation becomes

$$0 = -2r\dot{m}_{\text{nozzle}}V_r\cos\theta \quad \rightarrow \quad V_r = 0 \quad \rightarrow \quad V_{\text{jet,t}} - V_{\text{nozzle}} = 0$$

Noting that the tangential component of jet velocity is $V_{\text{jet,t}} = V_{\text{jet}}\cos\theta$, we have

$$V_{\text{nozzle}} = V_{\text{jet}}\cos\theta = (95.49\text{ m/s})\cos\theta$$

Also noting that $V_{\text{nozzle}} = \omega r = 2\pi\dot{n}r$, and angular speed and the rate of rotation of sprinkler head become

1) θ = 0°: $\omega = \frac{V_{\text{nozzle}}}{r} = \frac{(95.49\text{ m/s})\cos 0}{0.45\text{ m}} = \mathbf{212\text{ rad/s}}$ and $\dot{n} = \frac{\omega}{2\pi} = \frac{212\text{ rad/s}}{2\pi}\left(\frac{60\text{ s}}{1\text{ min}}\right) = 2026\text{ rpm} \cong \mathbf{2030\text{ rpm}}$

2) θ = 30°: $\omega = \frac{V_{\text{nozzle}}}{r} = \frac{(95.49\text{ m/s})\cos 30°}{0.45\text{ m}} = \mathbf{184\text{ rad/s}}$ and $\dot{n} = \frac{\omega}{2\pi} = \frac{184\text{ rad/s}}{2\pi}\left(\frac{60\text{ s}}{1\text{ min}}\right) = 1755\text{ rpm} \cong \mathbf{1760\text{ rpm}}$

3) θ = 60°: $\omega = \frac{V_{\text{nozzle}}}{r} = \frac{(95.49\text{ m/s})\cos 60°}{0.45\text{ m}} = \mathbf{106\text{ rad/s}}$ and $\dot{n} = \frac{\omega}{2\pi} = \frac{106\text{ rad/s}}{2\pi}\left(\frac{60\text{ s}}{1\text{ min}}\right) = 1013\text{ rpm} \cong \mathbf{1010\text{ rpm}}$

***Discussion*** Final results are given to three significant digits, as usual. The rate of rotation in reality will be lower because of frictional effects and air drag.



## 6-74

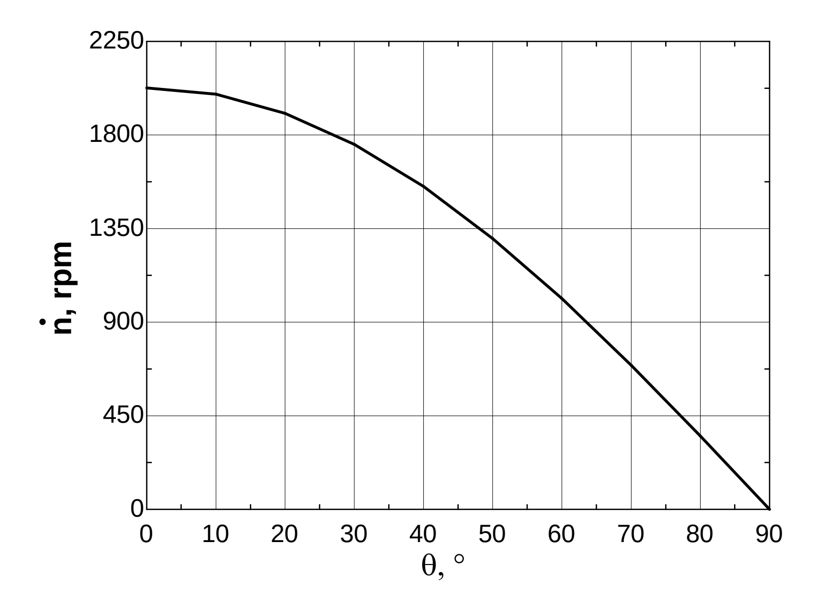


**Solution** The previous problem is reconsidered. The effect of discharge angle $\theta$ on the rate of rotation $\dot{n}$ as $\theta$ varies from 0 to 90° in increments of 10° is to be investigated.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
D=0.02 "m"
r=0.45 "m"
n_nozzle=2 "number of nozzles"
Ac=pi*D^2/4
V_jet=V_dot/Ac/n_nozzle
V_nozzle=V_jet*cos(theta)
V_dot=0.060 "m3/s"
omega=V_nozzle/r
n_dot=omega*60/(2*pi)
```

| Angle, $\theta$° | $V_{nozzle}$, m/s | ω rad/s | $\dot{n}$ rpm |
|---|---|---|---|
| 0 | 95.5 | 212 | 2026 |
| 10 | 94.0 | 209 | 1996 |
| 20 | 89.7 | 199 | 1904 |
| 30 | 82.7 | 184 | 1755 |
| 40 | 73.2 | 163 | 1552 |
| 50 | 61.4 | 136 | 1303 |
| 60 | 47.7 | 106 | 1013 |
| 70 | 32.7 | 73 | 693 |
| 80 | 16.6 | 37 | 352 |
| 90 | 0.0 | 0 | 0 |




***Discussion*** The maximum rpm occurs when $\theta = 0^o$, as expected, since this represents purely tangential outflow. When $\theta = 90^o$, the rpm drops to zero, as also expected, since the outflow is purely radial and therefore there is no torque to spin the sprinkler.

**6-75**

**Solution** A stationary water tank placed on wheels on a frictionless surface is propelled by a water jet that leaves the tank through a smooth hole. Relations are to be developed for the acceleration, the velocity, and the distance traveled by the tank as a function of time as water discharges.

***Assumptions*** **1** The orifice has a smooth entrance, and thus the frictional losses are negligible. **2** The flow is steady, incompressible, and irrotational (so that the Bernoulli equation is applicable). **3** The surface under the wheeled tank is level and frictionless. **4** The water jet is discharged horizontally and rearward. **5** The mass of the tank and wheel assembly is negligible compared to the mass of water in the tank. **4** Jet flow is nearly uniform and thus the momentum-flux correction factor can be taken to be unity, $\beta \cong 1$.

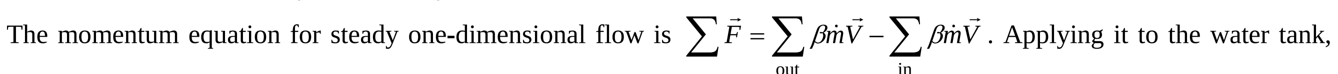
***Analysis*** (*a*) We take point 1 at the free surface of the tank, and point 2 at the outlet of the hole, which is also taken to be the reference level ($z_2 = 0$) so that the water height above the hole at any time is $z$. Noting that the fluid velocity at the free surface is very low ($V_1 \cong 0$), it is open to the atmosphere ($P_1 = P_{atm}$), and water discharges into the atmosphere (and thus $P_2 = P_{atm}$), the Bernoulli equation simplifies to

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad z=\frac{V_J^2}{2g}+0 \quad\rightarrow\quad V_J=\sqrt{2gz}$$



The discharge rate of water from the tank through the hole is

$$\dot{m}=\rho A V_J=\rho\frac{\pi D_0^2}{4}V_J=\rho\frac{\pi D_0^2}{4}\sqrt{2gz}$$



The momentum equation for steady one-dimensional flow is $\sum \vec{F}=\sum_{out}\beta\dot{m}\vec{V}-\sum_{in}\beta\dot{m}\vec{V}$. Applying it to the water tank, the horizontal force that acts on the tank is determined to be

$$F=\dot{m}V_e-0=\dot{m}V_J=\rho\frac{\pi D_0^2}{4}2gz=\rho gz\frac{\pi D_0^2}{2}$$

The acceleration of the water tank is determined from Newton's 2nd law of motion $F = ma$ where $m$ is the mass of water in the tank, $m=\rho V_{tank}=\rho(\pi D^2/4)z$,

$$a=\frac{F}{m}=\frac{\rho gz\left(\pi D_0^2/2\right)}{\rho z\left(\pi D^2/4\right)} \quad\rightarrow\quad \boxed{a=2g\frac{D_0^2}{D^2}}$$

Note that the acceleration of the tank is constant.

(*b*) Noting that $a = dV/dt$ and thus $dV = adt$ and acceleration $a$ is constant, the velocity is expressed as

$$V=at \quad\rightarrow\quad \boxed{V=2g\frac{D_0^2}{D^2}t}$$

(*c*) Noting that $V = dx/dt$ and thus $dx = Vdt$, the distance traveled by the water tank is determined by integration to be

$$dx=Vdt \quad\rightarrow\quad dx=2g\frac{D_0^2}{D^2}tdt \quad\rightarrow\quad \boxed{x=g\frac{D_0^2}{D^2}t^2}$$

since $x = 0$ at $t = 0$.

***Discussion*** In reality, the flow rate discharge velocity and thus the force acting on the water tank will be less because of the frictional losses at the hole. But these losses can be accounted for by incorporating a discharge coefficient.

---

## Design and Essay Problems

---

**6-76**

**Solution** Students' essays and designs should be unique and will differ from each other.

**Solutions Manual for**

**Fluid Mechanics: Fundamentals and Applications**

**by Çengel & Cimbala**

---

# CHAPTER 7
# DIMENSIONAL ANALYSIS AND MODELING

---

## Dimensions and Units, Primary Dimensions

### 7-1C
**Solution** We are to explain the difference between a dimension and a unit, and give examples.

*Analysis* **A dimension is a measure of a physical quantity (without numerical values), while a unit is a way to assign a number to that dimension**. Examples are numerous – length and meter, temperature and $^{o}$C, weight and lbf, mass and kg, time and second, power and watt,…

***Discussion*** When performing dimensional analysis, it is important to recognize the difference between dimensions and units.

### 7-2C
**Solution** We are to append the given table with other parameters and their primary dimensions.

***Analysis*** Students' tables will differ, but they should add entries such as angular velocity, kinematic viscosity, work, energy, power, specific heat, thermal conductivity, torque or moment, stress, etc.

***Discussion*** This problem should be assigned as an ongoing homework problem throughout the semester, and then collected towards the end. Individual instructors can determine how many entries to be required in the table.

### 7-3C
**Solution** We are to list the seven primary dimensions and explain their significance.

*Analysis* The seven primary dimensions are **mass, length, time, temperature, electrical current, amount of light, and amount of matter**. Their significance is that **all other dimensions can be formed by combinations of these seven primary dimensions**.

***Discussion*** One of the first steps in a dimensional analysis is to write down the primary dimensions of every variable or parameter that is important in the problem.

### 7-4
**Solution** We are to write the primary dimensions of the universal ideal gas constant.

*Analysis* From the given equation,

*Primary dimensions of the universal ideal gas constant*:

$$\{R_u\} = \left\{\frac{\text{pressure}\times\text{volume}}{\text{mol}\times\text{temperature}}\right\} = \left\{\frac{\frac{\text{m}}{\text{t}^2\text{L}}\times\text{L}^3}{\text{N}\times\text{T}}\right\} = \left\{\frac{\mathbf{mL^2}}{\mathbf{t^2TN}}\right\}$$

Or, in exponent form, $\{R_u\}$ = $\mathbf{\{m^1\,L^2\,t^{-2}\,T^{-1}\,N^{-1}\}}$.

***Discussion*** The standard value of $R_u$ is 8314.3 J/kmol·K. You can verify that these units agree with the dimensions of the result.

**7-5**

**Solution** We are to write the primary dimensions of atomic weight.

***Analysis*** By definition, atomic weight is mass per mol,

*Primary dimensions of atomic weight*:

$$\{M\} = \left\{\frac{\text{mass}}{\text{mol}}\right\} = \left\{\frac{\mathbf{m}}{\mathbf{N}}\right\} \qquad (1)$$

Or, in exponent form, $\{M\}$ = $\mathbf{\{m^1\,N^{-1}\}}$.

***Discussion*** In terms of primary dimensions, atomic mass is *not* dimensionless, although many authors treat it as such. Note that mass and amount of matter are defined as two separate primary dimensions.

---

**7-6**

**Solution** We are to write the primary dimensions of the universal ideal gas constant in the alternate system where force replaces mass as a primary dimension.

***Analysis*** From Newton's second law, force equals mass times acceleration. Thus, mass is written in terms of force as

*Primary dimensions of mass in the alternate system*:

$$\{\text{mass}\} = \left\{\frac{\text{force}}{\text{acceleration}}\right\} = \left\{\frac{\text{F}}{\text{L/t}^2}\right\} = \left\{\frac{\text{Ft}^2}{\text{L}}\right\} \qquad (1)$$

We substitute Eq. 1 into the results of Problem 7-4,

*Primary dimensions of the universal ideal gas constant*:

$$\{R_u\} = \left\{\frac{\text{mL}^2}{\text{Nt}^2\text{T}}\right\} = \left\{\frac{\frac{\text{Ft}^2}{\text{L}}\text{L}^2}{\text{Nt}^2\text{T}}\right\} = \left\{\frac{\mathbf{FL}}{\mathbf{TN}}\right\} \qquad (2)$$

Or, in exponent form, $\{R_u\}$ = $\mathbf{\{F^1\,L^1\,T^{-1}\,N^{-1}\}}$.

***Discussion*** The standard value of $R_u$ is 8314.3 J/kmol·K. You can verify that these units agree with the dimensions of Eq. 2.

---

**7-7**

**Solution** We are to write the primary dimensions of the specific ideal gas constant, and verify are result by comparing to the standard SI units of $R_{air}$.

***Analysis*** We can approach this problem two ways. If we have already worked through Problem 7-4, we can use our results. Namely,

*Primary dimensions of specific ideal gas constant*:

$$\{R_{gas}\} = \left\{\frac{R_u}{M}\right\} = \left\{\frac{\frac{mL^2}{Nt^2T}}{\frac{m}{N}}\right\} = \left\{\frac{\mathbf{L^2}}{\mathbf{t^2T}}\right\} \qquad (1)$$

Or, in exponent form, $\{R_{gas}\}$ = $\mathbf{\{L^2\, t^{-2}\, T^{-1}\}}$. Alternatively, we can use either form of the ideal gas law,

*Primary dimensions of specific ideal gas constant*:

$$\{R_{gas}\} = \left\{\frac{\text{pressure}\times\text{volume}}{\text{mass}\times\text{temperature}}\right\} = \left\{\frac{\frac{m}{t^2L}\times L^3}{m\times T}\right\} = \left\{\frac{\mathbf{L^2}}{\mathbf{t^2T}}\right\} \qquad (2)$$

For air, $R_{air}$ = 287.0 J/kg·K. We transform these units into primary dimensions,

*Primary dimensions of the specific ideal gas constant for air*:

$$\{R_{air}\} = \left\{287.0\frac{J}{kg\times K}\right\} = \left\{\frac{\frac{mL^2}{t^2}}{m\times T}\right\} = \left\{\frac{\mathbf{L^2}}{\mathbf{t^2T}}\right\} \qquad (3)$$

Equation 3 agrees with Eq. 1 and Eq. 2, increasing our confidence that we have performed the algebra correctly.

***Discussion*** Notice that numbers, like the value 287.0 in Eq. 3 have no influence on the dimensions.

---

**7-8**

**Solution** We are to write the primary dimensions of torque and list its units.

***Analysis*** Torque is the product of a length and a force,

*Primary dimensions of torque*: $$\{\vec{M}\} = \left\{\text{length}\times\text{mass}\frac{\text{length}}{\text{time}^2}\right\} = \left\{\mathbf{m}\frac{\mathbf{L^2}}{\mathbf{t^2}}\right\} \qquad (1)$$

Or, in exponent form, $\{\vec{M}\}$ = $\mathbf{\{m^1\ L^2\ t^{-2}\}}$. The common units of torque are newton-meter (SI) and inch-pound (English). In *primary units*, however, we write the primary SI units according to Eq. 1,

*Primary SI units*: Units of torque = $\mathbf{kg\cdot m^2/s^2}$

and in primary English units,

*Primary English units*: Units of torque = $\mathbf{lbm\cdot ft^2/s^2}$

***Discussion*** Since torque is the product of a force and a length, it has the same dimensions as energy. Primary *units* are not required for dimensional analysis, but are often useful for unit conversions and for verification of proper units when solving a problem.

---

**7-9**

**Solution** We are to determine the primary dimensions of each variable.

***Analysis***

(*a*) Energy is force times length (the same dimensions as work),

*Primary dimensions of energy*:

$$\{E\} = \{\text{force} \times \text{length}\} = \left\{\frac{\text{mass} \times \text{length}}{\text{time}^2} \times \text{length}\right\} = \left\{\frac{\mathbf{mL^2}}{\mathbf{t^2}}\right\} \qquad (1)$$

Or, in exponent form, $\{E\}$ = $\{\mathbf{m^1 L^2 t^{-2}}\}$.

(*b*) Specific energy is energy per unit mass,

*Primary dimensions of specific energy*:

$$\{e\} = \left\{\frac{\text{energy}}{\text{mass}}\right\} = \left\{\frac{\text{mass} \times \text{length}^2}{\text{time}^2} \times \frac{1}{\text{mass}}\right\} = \left\{\frac{\mathbf{L^2}}{\mathbf{t^2}}\right\} \qquad (2)$$

Or, in exponent form, $\{e\}$ = $\{\mathbf{L^2 t^{-2}}\}$.

(*c*) Power is the rate of change of energy, i.e. energy per unit time,

*Primary dimensions of power*:

$$\{\dot{W}\} = \left\{\frac{\text{energy}}{\text{time}}\right\} = \left\{\frac{\text{mass} \times \text{length}^2}{\text{time}^2} \times \frac{1}{\text{time}}\right\} = \left\{\frac{\mathbf{mL^2}}{\mathbf{t^3}}\right\} \qquad (3)$$

Or, in exponent form, $\{\dot{W}\}$ = $\{\mathbf{m^1 L^2 t^{-3}}\}$.

***Discussion*** In dimensional analysis it is important to distinguish between energy, specific energy, and power.

---

**7-10**

**Solution** We are to determine the primary dimensions of electrical voltage.

***Analysis*** From the hint,

*Primary dimensions of voltage*:

$$\{\text{voltage}\} = \left\{\frac{\text{power}}{\text{current}}\right\} = \left\{\frac{\dfrac{\text{mass} \times \text{length}^2}{\text{time}^3}}{\text{current}}\right\} = \left\{\frac{\mathbf{mL^2}}{\mathbf{t^3 I}}\right\} \qquad (1)$$

Or, in exponent form, $\{E\}$ = $\{\mathbf{m^1 L^2 t^{-3} I^{-1}}\}$.

***Discussion*** We see that all dimensions, even those of electrical properties, can be expressed in terms of primary dimensions.

---

**7-11**

**Solution** We are to write the primary dimensions of electrical resistance.

***Analysis*** From Ohm's law, we see that resistance has the dimensions of voltage difference divided by electrical current,

*Primary dimensions of resistance*:

$$\{R\} = \left\{\frac{\Delta E}{I}\right\} = \left\{\frac{\dfrac{\text{mass} \times \text{length}^2}{\text{time}^3 \times \text{current}}}{\text{current}}\right\} = \left\{\frac{\mathbf{mL^2}}{\mathbf{t^3 I^2}}\right\}$$

Or, in exponent form, $\{R\}$ = $\{\mathbf{m^1\, L^2\, t^{-3}\, I^{-2}}\}$, where we have also used the result of the previous problem.

***Discussion*** *All* dimensions can be written in terms of primary dimensions.

---

**7-12**

**Solution** We are to determine the primary dimensions of each variable.

***Analysis***

(*a*) Acceleration is the rate of change of velocity,

*Primary dimensions of acceleration*:

$$\{a\} = \left\{\frac{\text{velocity}}{\text{time}}\right\} = \left\{\frac{\text{length}}{\text{time}} \times \frac{1}{\text{time}}\right\} = \left\{\frac{\mathbf{L}}{\mathbf{t^2}}\right\} \qquad (1)$$

Or, in exponent form, $\{a\}$ = $\{\mathbf{L^1\, t^{-2}}\}$.

(*b*) Angular velocity is the rate of change of angle,

*Primary dimensions of angular velocity*: $$\{\omega\} = \left\{\frac{\text{angle}}{\text{time}}\right\} = \left\{\frac{1}{\text{time}}\right\} = \left\{\frac{\mathbf{1}}{\mathbf{t}}\right\} \qquad (2)$$

Or, in exponent form, $\{\omega\}$ = $\{\mathbf{t^{-1}}\}$.

(*c*) Angular acceleration is the rate of change of angular velocity,

*Primary dimensions of angular acceleration*:

$$\{\alpha = \dot{\omega}\} = \left\{\frac{\text{angular velocity}}{\text{time}}\right\} = \left\{\frac{1}{\text{time}} \times \frac{1}{\text{time}}\right\} = \left\{\frac{\mathbf{1}}{\mathbf{t^2}}\right\} \qquad (3)$$

Or, in exponent form, $\{\alpha\}$ = $\{\mathbf{t^{-2}}\}$.

***Discussion*** In Part (*b*) we note that the unit of angle is radian, which is a dimensionless unit. Therefore the dimensions of angle are unity.

---

**7-13**

**Solution** We are to write the primary dimensions of angular momentum and list its units.

***Analysis*** Angular momentum is the product of length, mass, and velocity,

*Primary dimensions of angular momentum*:

$$\left\{\vec{H}\right\} = \left\{\text{length}\times\text{mass}\times\frac{\text{length}}{\text{time}}\right\} = \left\{\frac{\mathbf{mL^2}}{\mathbf{t}}\right\} \qquad (1)$$

Or, in exponent form, { $\vec{H}$ } = $\{\mathbf{m^1 L^2 t^{-1}}\}$. We write the primary SI units according to Eq. 1,

*Primary SI units*: Units of angular momentum $= \dfrac{\mathbf{kg\cdot m^2}}{\mathbf{s}}$

and in primary English units,

*Primary English units*: Units of angular momentum $= \dfrac{\mathbf{lbm\cdot ft^2}}{\mathbf{s}}$

***Discussion*** Primary units are not required for dimensional analysis, but are often useful for unit conversions and for verification of proper units when solving a problem.

---

**7-14**

**Solution** We are to determine the primary dimensions of each variable.

***Analysis***
(*a*) Specific heat is energy per unit mass per unit temperature,

*Primary dimensions of specific heat at constant pressure*:

$$\left\{c_p\right\} = \left\{\frac{\text{energy}}{\text{mass}\times\text{temperature}}\right\} = \left\{\frac{\frac{\text{mL}^2}{\text{t}^2}}{\text{m}\times\text{T}}\right\} = \left\{\frac{\mathbf{L^2}}{\mathbf{t^2T}}\right\} \qquad (1)$$

Or, in exponent form, $\{c_p\}$ = $\{\mathbf{L^2 t^{-2} T^{-1}}\}$.

(*b*) Specific weight is density times gravitational acceleration,

*Primary dimensions of specific weight*: $\left\{\rho g\right\} = \left\{\frac{\text{mass}}{\text{volume}}\frac{\text{length}}{\text{time}^2}\right\} = \left\{\frac{\mathbf{m}}{\mathbf{L^2t^2}}\right\} \qquad (2)$

Or, in exponent form, $\{\rho g\}$ = $\{\mathbf{m^1 L^{-2} t^{-2}}\}$.

(*c*) Specific enthalpy has dimensions of energy per unit mass,

*Primary dimensions of specific enthalpy*: $\left\{h\right\} = \left\{\frac{\text{energy}}{\text{mass}}\right\} = \left\{\frac{\frac{\text{mL}^2}{\text{t}^2}}{\text{m}}\right\} = \left\{\frac{\mathbf{L^2}}{\mathbf{t^2}}\right\} \qquad (3)$

Or, in exponent form, $\{h\}$ = $\{\mathbf{L^2 t^{-2}}\}$.

***Discussion*** As a check, from our study of thermodynamics we know that $dh = c_p dT$ for an ideal gas. Thus, the dimensions of $dh$ must equal the dimensions of $c_p$ times the dimensions of $dT$. Comparing Eqs. 1 and 3 above, we see that this is indeed the case.

---

**7-15**

**Solution** We are to determine the primary dimensions of thermal conductivity.

***Analysis*** The primary dimensions of $\dot{Q}_{\text{conduction}}$ are energy/time, and the primary dimensions of $dT/dx$ are temperature/length. From the given equation,

*Primary dimensions of thermal conductivity*:

$$\{k\} = \left\{ \frac{\dfrac{\text{energy}}{\text{time}}}{\text{length}^2 \times \dfrac{\text{temperature}}{\text{length}}} \right\} = \left\{ \frac{\dfrac{\text{mL}^2}{\text{t}^3}}{\text{L} \times \text{T}} \right\} = \left\{ \frac{\mathbf{mL}}{\mathbf{t^3T}} \right\} \quad (1)$$

Or, in exponent form, $\{k\}$ = $\{\mathbf{m^1\,L^1\,t^{-3}\,T^{-1}}\}$. We obtain a value of $k$ from a reference book. E.g. $k_{\text{copper}}$ = 401 W/m·K. These units have dimensions of power/length·temperature. Since power is energy/time, we see immediately that Eq. 1 is correct. Alternatively, we can transform the units of $k$ into primary units,

*Primary SI units of thermal conductivity*:

$$k_{\text{copper}} = 401 \frac{\text{W}}{\text{m K}} \left( \frac{\text{N m}}{\text{s W}} \right) \left( \frac{\text{kg m}}{\text{N s}^2} \right) = 401 \frac{\mathbf{kg \cdot m}}{\mathbf{s^3 \cdot K}} \quad (2)$$

***Discussion*** We have used the principle of dimensional homogeneity to determine the primary dimensions of $k$. Namely, we utilized the fact that the dimensions of both terms of the given equation must be identical.

---

**7-16**

**Solution** We are to determine the primary dimensions of each variable.

***Analysis***

(*a*) Heat generation rate is energy per unit volume per unit time,

*Primary dimensions of heat generation rate*:

$$\{\dot{g}\} = \left\{ \frac{\text{energy}}{\text{volume} \times \text{time}} \right\} = \left\{ \frac{\dfrac{\text{mL}^2}{\text{t}^2}}{\text{L}^3\text{t}} \right\} = \left\{ \frac{\mathbf{m}}{\mathbf{Lt^3}} \right\} \quad (1)$$

Or, in exponent form, $\{\dot{g}\}$ = $\{\mathbf{m^1\,L^{-1}\,t^{-3}}\}$.

(*b*) Heat flux is energy per unit area per unit time,

*Primary dimensions of heat flux*:

$$\{\dot{q}\} = \left\{ \frac{\text{energy}}{\text{area} \times \text{time}} \right\} = \left\{ \frac{\dfrac{\text{mL}^2}{\text{t}^2}}{\text{L}^2\text{t}} \right\} = \left\{ \frac{\mathbf{m}}{\mathbf{t^3}} \right\} \quad (2)$$

Or, in exponent form, $\{\dot{q}\}$ = $\{\mathbf{m^1\,t^{-3}}\}$.

(*c*) Heat flux is energy per unit area per unit time per unit temperature,

*Primary dimensions of heat transfer coefficient*:

$$\{h\} = \left\{ \frac{\text{energy}}{\text{area} \times \text{time} \times \text{temperature}} \right\} = \left\{ \frac{\dfrac{\text{mL}^2}{\text{t}^2}}{\text{L}^2 \times \text{t} \times \text{T}} \right\} = \left\{ \frac{\mathbf{m}}{\mathbf{t^3T}} \right\} \quad (3)$$

Or, in exponent form, $\{h\}$ = $\{\mathbf{m^1\,t^{-3}\,T^{-1}}\}$.

***Discussion*** In the field of heat transfer it is critical that one be careful with the dimensions (and units) of heat transfer variables.

---

**7-17**

**Solution** We are to choose three properties or constants and write out their names, their SI units, and their primary dimensions.

***Analysis*** There are many options. For example,

Students may choose $c_v$ (specific heat at constant volume). The units are kJ/kg·K, which is energy per mass per temperature. Thus,

*Primary dimensions of specific heat at constant volume*:

$$\{c_v\} = \left\{\frac{\text{energy}}{\text{mass} \times \text{temperature}}\right\} = \left\{\frac{\frac{\text{mL}^2}{\text{t}^2}}{\text{m} \times \text{T}}\right\} = \left\{\frac{\mathbf{L^2}}{\mathbf{t^2 T}}\right\} \qquad (1)$$

Or, in exponent form, $\{c_v\}$ = $\{\mathbf{L^2\, t^{-2}\, T^{-1}}\}$.

Students may choose $v$ (specific volume). The units are m$^3$/kg, which is volume per mass. Thus,

*Primary dimensions of specific volume*:

$$\{v\} = \left\{\frac{\text{volume}}{\text{mass}}\right\} = \left\{\frac{\mathbf{L^3}}{\mathbf{m}}\right\} \qquad (2)$$

Or, in exponent form, $\{v\}$ = $\{\mathbf{m^{-1}\, L^3}\}$.

Students may choose $h_{fg}$ (latent heat of vaporization). The units are kJ/kg, which is energy per mass. Thus,

*Primary dimensions of latent heat of vaporization*:

$$\{h_{fg}\} = \left\{\frac{\text{energy}}{\text{mass}}\right\} = \left\{\frac{\frac{\text{mL}^2}{\text{t}^2}}{\text{m}}\right\} = \left\{\frac{\mathbf{L^2}}{\mathbf{t^2}}\right\} \qquad (3)$$

Or, in exponent form, $\{h_{fg}\}$ = $\{\mathbf{L^2\, t^{-2}}\}$. (The same dimensions hold for $h_f$ and $h_g$.)

Students may choose $s_f$ (specific entropy of a saturated liquid). The units are kJ/kg·K, which is energy per mass per temperature. Thus,

*Primary dimensions of specific entropy of a saturated liquid*:

$$\{s_f\} = \left\{\frac{\text{energy}}{\text{mass} \times \text{temperature}}\right\} = \left\{\frac{\frac{\text{mL}^2}{\text{t}^2}}{\text{m} \times \text{T}}\right\} = \left\{\frac{\mathbf{L^2}}{\mathbf{t^2 T}}\right\} \qquad (4)$$

Or, in exponent form, $\{s_f\}$ = $\{\mathbf{L^2\, t^{-2}\, T^{-1}}\}$. (The same dimensions hold for $s_{fg}$ and $s_g$.)

***Discussion*** Students' answers will vary. There are some other choices.

**7-18E**

**Solution** We are to choose three properties or constants and write out their names, their SI units, and their primary dimensions.

***Analysis*** There are many options. For example,

Students may choose $c_v$ (specific heat at constant volume). The units are Btu/lbm·R, which is energy per mass per temperature. Thus,

*Primary dimensions of specific heat at constant volume*:

$$\{c_v\} = \left\{\frac{\text{energy}}{\text{mass} \times \text{temperature}}\right\} = \left\{\frac{\frac{\text{mL}^2}{\text{t}^2}}{\text{m} \times \text{T}}\right\} = \left\{\frac{\mathbf{L^2}}{\mathbf{t^2 T}}\right\} \qquad (1)$$

Or, in exponent form, $\{c_v\}$ = $\{\mathbf{L^2\, t^{-2}\, T^{-1}}\}$.

Students may choose $v$ (specific volume). The units are $\text{ft}^3$/lbm, which is volume per mass. Thus,

*Primary dimensions of specific volume*:

$$\{v\} = \left\{\frac{\text{volume}}{\text{mass}}\right\} = \left\{\frac{\mathbf{L^3}}{\mathbf{m}}\right\} \qquad (2)$$

Or, in exponent form, $\{v\}$ = $\{\mathbf{m^{-1}\, L^3}\}$.

Students may choose $h_{fg}$ (latent heat of vaporization). The units are Btu/lbm, which is energy per mass. Thus,

*Primary dimensions of latent heat of vaporization*:

$$\{h_{fg}\} = \left\{\frac{\text{energy}}{\text{mass}}\right\} = \left\{\frac{\frac{\text{mL}^2}{\text{t}^2}}{\text{m}}\right\} = \left\{\frac{\mathbf{L^2}}{\mathbf{t^2}}\right\} \qquad (3)$$

Or, in exponent form, $\{h_{fg}\}$ = $\{\mathbf{L^2\, t^{-2}}\}$. (The same dimensions hold for $h_f$ and $h_g$.)

Students may choose $s_f$ (specific entropy of a saturated liquid). The units are Btu/lbm·R, which is energy per mass per temperature. Thus,

*Primary dimensions of specific entropy of a saturated liquid*:

$$\{s_f\} = \left\{\frac{\text{energy}}{\text{mass} \times \text{temperature}}\right\} = \left\{\frac{\frac{\text{mL}^2}{\text{t}^2}}{\text{m} \times \text{T}}\right\} = \left\{\frac{\mathbf{L^2}}{\mathbf{t^2 T}}\right\} \qquad (4)$$

Or, in exponent form, $\{s_f\}$ = $\{\mathbf{L^2\, t^{-2}\, T^{-1}}\}$. (The same dimensions hold for $s_{fg}$ and $s_g$.)

***Discussion*** Students' answers will vary. There are some other choices.

## Dimensional Homogeneity

### 7-19C

**Solution** We are to explain the law of dimensional homogeneity.

***Analysis*** The law of dimensional homogeneity states that **every additive term in an equation must have the same dimensions**. As a simple counter example, an equation with one term of dimensions length and another term of dimensions temperature would clearly violate the law of dimensional homogeneity – you cannot add length and temperature. All terms in the equation must have the *same* dimensions.

***Discussion*** If in the solution of an equation you realize that the dimensions of two terms are not equivalent, this is a sure sign that you have made a mistake somewhere!

### 7-20

**Solution** We are to determine the primary dimensions of the gradient operator, and then verify that primary dimensions of each additive term in the equation are the same.

***Analysis***

(*a*) By definition, the gradient operator is a three-dimensional derivative operator. For example, in Cartesian coordinates,

*Gradient operator in Cartesian coordinates*:

$$\vec{\nabla} = \left( \frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \right) = \vec{i}\,\frac{\partial}{\partial x} + \vec{j}\,\frac{\partial}{\partial y} + \vec{k}\,\frac{\partial}{\partial z}$$

Therefore its dimensions must be 1/length. Thus,

*Primary dimensions of the gradient operator*: $\left\{\vec{\nabla}\right\} = \left\{\frac{\mathbf{1}}{\mathbf{L}}\right\}$

Or, in exponent form, $\{\vec{\nabla}\} = \{\mathbf{L}^{-1}\}$.

(*b*) Similarly, the primary dimensions of a time derivative ($\partial/\partial t$) are 1/time. Also, the primary dimensions of velocity are length/time, and the primary dimensions of acceleration are length/time$^2$. Thus each term in the given equation can be written in terms of primary dimensions,

$$\{\vec{a}\} = \left\{\frac{\text{length}}{\text{time}^2}\right\} \qquad \{\vec{a}\} = \left\{\frac{\text{L}}{\text{t}^2}\right\}$$

$$\left\{\frac{\partial \vec{V}}{\partial t}\right\} = \left\{\frac{\frac{\text{length}}{\text{time}}}{\text{time}}\right\} = \left\{\frac{\text{length}}{\text{time}^2}\right\} \qquad \left\{\frac{\partial \vec{V}}{\partial t}\right\} = \left\{\frac{\text{L}}{\text{t}^2}\right\}$$

$$\left\{\left(\vec{V}\cdot\vec{\nabla}\right)\vec{V}\right\} = \left\{\frac{\text{length}}{\text{time}} \times \frac{1}{\text{length}} \times \frac{\text{length}}{\text{time}}\right\} = \left\{\frac{\text{length}}{\text{time}^2}\right\} \qquad \left\{\left(\vec{V}\cdot\vec{\nabla}\right)\vec{V}\right\} = \left\{\frac{\text{L}}{\text{t}^2}\right\}$$

Indeed, **all three additive terms have the same dimensions, namely $\{\mathbf{L}^{1}\,\mathbf{t}^{-2}\}$**.

***Discussion*** If the dimensions of any of the terms were different from the others, it would be a sure sign that an error was made somewhere in deriving or copying the equation.

**7-21**

**Solution** We are to determine the primary dimensions of each additive term in the equation, and we are to verify that the equation is dimensionally homogeneous.

***Analysis*** The primary dimensions of the time derivative ($\partial/\partial t$) are 1/time. The primary dimensions of the gradient vector are 1/length, and the primary dimensions of velocity are length/time. Thus each term in the equation can be written in terms of primary dimensions,

$$\left\{\frac{\vec{F}}{m}\right\} = \left\{\frac{\text{force}}{\text{mass}}\right\} = \left\{\frac{\frac{\text{mass}\times\text{length}}{\text{time}^2}}{\text{mass}}\right\} \qquad \left\{\frac{\vec{F}}{m}\right\} = \left\{\frac{\text{L}}{\text{t}^2}\right\}$$

$$\left\{\frac{\partial\vec{V}}{\partial t}\right\} = \left\{\frac{\frac{\text{length}}{\text{time}}}{\text{time}}\right\} \qquad \left\{\frac{\partial\vec{V}}{\partial t}\right\} = \left\{\frac{\text{L}}{\text{t}^2}\right\}$$

$$\left\{\left(\vec{V}\cdot\vec{\nabla}\right)\vec{V}\right\} = \left\{\frac{\text{length}}{\text{time}}\times\frac{1}{\text{length}}\times\frac{\text{length}}{\text{time}}\right\} \qquad \left\{\left(\vec{V}\cdot\vec{\nabla}\right)\vec{V}\right\} = \left\{\frac{\text{L}}{\text{t}^2}\right\}$$

Indeed, **all three additive terms have the same dimensions, namely $\{L^1\, t^{-2}\}$**.

***Discussion*** The dimensions are, in fact, those of acceleration.

---

**7-22**

**Solution** We are to determine the primary dimensions of each additive term in Eq. 1, and we are to verify that the equation is dimensionally homogeneous.

***Analysis*** The primary dimensions of the material derivative ($D/Dt$) are 1/time. The primary dimensions of volume are length$^3$, and the primary dimensions of velocity are length/time. Thus each term in the equation can be written in terms of primary dimensions,

$$\left\{\frac{1}{V}\frac{DV}{Dt}\right\} = \left\{\frac{1}{\text{length}^3}\times\frac{\text{length}^3}{\text{time}}\right\} = \left\{\frac{1}{\text{time}}\right\} \qquad \left\{\frac{1}{V}\frac{DV}{Dt}\right\} = \left\{\frac{1}{\text{t}}\right\}$$

$$\left\{\frac{\partial u}{\partial x}\right\} = \left\{\frac{\frac{\text{length}}{\text{time}}}{\text{length}}\right\} = \left\{\frac{1}{\text{time}}\right\} \qquad \left\{\frac{\partial u}{\partial x}\right\} = \left\{\frac{1}{\text{t}}\right\}$$

$$\left\{\frac{\partial v}{\partial y}\right\} = \left\{\frac{\frac{\text{length}}{\text{time}}}{\text{length}}\right\} = \left\{\frac{1}{\text{time}}\right\} \qquad \left\{\frac{\partial v}{\partial y}\right\} = \left\{\frac{1}{\text{t}}\right\}$$

$$\left\{\frac{\partial w}{\partial z}\right\} = \left\{\frac{\frac{\text{length}}{\text{time}}}{\text{length}}\right\} = \left\{\frac{1}{\text{time}}\right\} \qquad \left\{\frac{\partial w}{\partial z}\right\} = \left\{\frac{1}{\text{t}}\right\}$$

Indeed, **all four additive terms have the same dimensions, namely $\{t^{-1}\}$**.

***Discussion*** If the dimensions of any of the terms were different from the others, it would be a sure sign that an error was made somewhere in deriving or copying the equation.

---

**7-23**

**Solution** We are to determine the primary dimensions of each additive term, and we are to verify that the equation is dimensionally homogeneous.

***Analysis*** The primary dimensions of the velocity components are length/time. The primary dimensions of coordinates *r* and *z* are length, and the primary dimensions of coordinate $\theta$ are unity (it is a dimensionless angle). Thus each term in the equation can be written in terms of primary dimensions,

$$\left\{\frac{1}{r}\frac{\partial(ru_r)}{\partial r}\right\} = \left\{\frac{1}{\text{length}} \times \frac{\text{length}\,\frac{\text{length}}{\text{time}}}{\text{length}}\right\} = \left\{\frac{1}{\text{time}}\right\} \qquad \left\{\frac{1}{r}\frac{\partial(ru_r)}{\partial r}\right\} = \left\{\frac{1}{\text{t}}\right\}$$

$$\left\{\frac{1}{r}\frac{\partial u_\theta}{\partial \theta}\right\} == \left\{\frac{1}{\text{length}} \times \frac{\frac{\text{length}}{\text{time}}}{1}\right\} = \left\{\frac{1}{\text{time}}\right\} \qquad \left\{\frac{1}{r}\frac{\partial u_\theta}{\partial \theta}\right\} = \left\{\frac{1}{\text{t}}\right\}$$

$$\left\{\frac{\partial u_z}{\partial z}\right\} = \left\{\frac{\frac{\text{length}}{\text{time}}}{\text{length}}\right\} = \left\{\frac{1}{\text{time}}\right\} \qquad \left\{\frac{\partial u_z}{\partial z}\right\} = \left\{\frac{1}{\text{t}}\right\}$$

Indeed, **all three additive terms have the same dimensions, namely {$t^{-1}$}**.

***Discussion*** If the dimensions of any of the terms were different from the others, it would be a sure sign that an error was made somewhere in deriving or copying the equation.

## 7-24

**Solution** We are to determine the primary dimensions of each additive term in the equation, and we are to verify that the equation is dimensionally homogeneous.

***Analysis*** The primary dimensions of heat transfer rate are energy/time. The primary dimensions of mass flow rate are mass/time, and those of specific heat are energy/mass·temperature, as found in Problem 7-14. Thus each term in the equation can be written in terms of primary dimensions,

$$\left\{\dot{Q}\right\} = \left\{\frac{\text{energy}}{\text{time}}\right\} = \left\{\frac{\frac{\text{mL}^2}{\text{t}^2}}{\text{t}}\right\} = \left\{\frac{\text{mL}^2}{\text{t}^3}\right\} \qquad \left\{\dot{Q}\right\} = \left\{\frac{\text{mL}^2}{\text{t}^3}\right\}$$

$$\left\{\dot{m}C_pT_{\text{out}}\right\} = \left\{\frac{\text{mass}}{\text{time}}\times\frac{\text{energy}}{\text{mass}\times\text{temperature}}\times\text{temperature}\right\} = \left\{\frac{\text{m}}{\text{t}}\times\frac{\frac{\text{mL}^2}{\text{t}^2}}{\text{m}\times\text{T}}\times\text{T}\right\} = \left\{\frac{\text{mL}^2}{\text{t}^3}\right\} \qquad \left\{\dot{m}C_pT_{\text{out}}\right\} = \left\{\frac{\text{mL}^2}{\text{t}^3}\right\}$$

$$\left\{\dot{m}C_pT_{\text{in}}\right\} = \left\{\frac{\text{mass}}{\text{time}}\times\frac{\text{energy}}{\text{mass}\times\text{temperature}}\times\text{temperature}\right\} = \left\{\frac{\text{m}}{\text{t}}\times\frac{\frac{\text{mL}^2}{\text{t}^2}}{\text{m}\times\text{T}}\times\text{T}\right\} = \left\{\frac{\text{mL}^2}{\text{t}^3}\right\} \qquad \left\{\dot{m}C_pT_{\text{in}}\right\} = \left\{\frac{\text{mL}^2}{\text{t}^3}\right\}$$

Indeed, **all three additive terms have the same dimensions, namely {$\mathbf{m^1\,L^2\,t^{-3}}$}**.

***Discussion*** We could also have left the temperature difference in parentheses as a temperature difference (same dimensions as the individual temperatures), and treated the equation as having only two terms.

---

## 7-25

**Solution** We are to determine the primary dimensions of each additive term, and we are to verify that the equation is dimensionally homogeneous.

***Analysis*** The primary dimensions of the time derivative (*d*/*dt*) are 1/time. The primary dimensions of density are mass/length$^3$, those of volume are length$^3$, those of area are length$^2$, and those of velocity are length/time. The primary dimensions of unit vector $\vec{n}$ are unity, i.e. {1} (in other words $\vec{n}$ has no dimensions). Finally, the primary dimensions of *b*, which is defined as *B* per unit mass, are {*B*/m}. Thus each term in the equation can be written in terms of primary dimensions,

$$\left\{\frac{dB_{\text{sys}}}{dt}\right\} = \left\{\frac{B}{\text{time}}\right\} \qquad \left\{\frac{dB_{\text{sys}}}{dt}\right\} = \left\{\frac{B}{\text{t}}\right\}$$

$$\left\{\frac{d}{dt}\int_{\text{CV}}\rho b dV\right\} = \left\{\frac{1}{\text{time}}\times\frac{\text{mass}}{\text{length}^3}\times\frac{B}{\text{mass}}\times\text{length}^3\right\} \qquad \left\{\frac{d}{dt}\int_{\text{CV}}\rho b dV\right\} = \left\{\frac{B}{\text{t}}\right\}$$

$$\left\{\int_{\text{CS}}\rho b\vec{V}_{\text{r}}\cdot\vec{n}dA\right\} = \left\{\frac{\text{mass}}{\text{length}^3}\times\frac{B}{\text{mass}}\times\frac{\text{length}}{\text{time}}\times 1\times\text{length}^2\right\} \qquad \left\{\int_{\text{CS}}\rho b\vec{V}_{\text{r}}\cdot\vec{n}dA\right\} = \left\{\frac{B}{\text{t}}\right\}$$

Indeed, **all three additive terms have the same dimensions, namely {$\boldsymbol{B}\,\mathbf{t^{-1}}$}**.

***Discussion*** The RTT for property *B* has dimensions of rate of change of *B*.

---

**7-26**

**Solution** We are to determine the primary dimensions of the first three additive term, and we are to verify that those terms are dimensionally homogeneous. Then we are to evaluate the dimensions of the adsorption coefficient.

***Analysis*** The primary dimensions of the time derivative (*d*/*dt*) are 1/time. Those of $A_s$ are length$^2$, those of $V$ are length$^3$, those of *c* are mass/length$^3$, and those of $\dot{V}$ are length$^3$/time. Thus the primary dimensions of the first three terms are

$$\left\{V\frac{dc}{dt}\right\} = \left\{\text{length}^3\frac{\frac{\text{mass}}{\text{length}^3}}{\text{time}}\right\} = \left\{\frac{\text{mass}}{\text{time}}\right\} \qquad \left\{V\frac{dc}{dt}\right\} = \left\{\frac{\text{m}}{\text{t}}\right\}$$

$$\{S\} = \left\{\frac{\text{mass}}{\text{time}}\right\} \qquad \{S\} = \left\{\frac{\text{m}}{\text{t}}\right\}$$

$$\{\dot{V}c\} = \left\{\frac{\text{length}^3}{\text{time}}\times\frac{\text{mass}}{\text{length}^3}\right\} = \left\{\frac{\text{mass}}{\text{time}}\right\} \qquad \{\dot{V}c\} = \left\{\frac{\text{m}}{\text{t}}\right\}$$

Indeed, the first three additive terms have the same dimensions, namely $\mathbf{\{m^1\,t^{-1}\}}$. Since the equation must be dimensionally homogeneous, the last term must have the same dimensions as well. We use this fact to find the dimensions of $k_w$,

$$\{cA_s k_w\} = \left\{\frac{\text{mass}}{\text{time}}\right\} \qquad \{k_w\} = \left\{\frac{\frac{\text{mass}}{\text{time}}}{cA_s}\right\} = \left\{\frac{\frac{\text{mass}}{\text{time}}}{\frac{\text{mass}}{\text{length}^3}\times\text{length}^2}\right\} \qquad \{k_w\} = \left\{\frac{\mathbf{L}}{\mathbf{t}}\right\}$$

Or, in exponent form, $\{k_w\} = \mathbf{\{L^1\,t^{-1}\}}$. The dimensions of wall adsorption coefficient are those of velocity.

***Discussion*** In fact, some authors call $k_w$ a "deposition velocity".

## Nondimensionalization of Equations

### 7-27C

**Solution** We are to give the primary reason for nondimensionalizing an equation.

***Analysis*** The primary reason for nondimensionalizing an equation is **to reduce the number of parameters in the problem**.

***Discussion*** As shown in the examples in the text, nondimensionalization of an equation reduces the number of independent parameters in the problem, simplifying the analysis.

### 7-28

**Solution** We are to nondimensionalize all the variables, and then re-write the equation in nondimensionalized form.

***Assumptions*** **1** The air in the room is well mixed so that $c$ is only a function of time.

***Analysis***
($a$) We nondimensionalize the variables by inspection according to their dimensions,

*Nondimensionalized variables*:

$$V^* = \frac{V}{L^3},\ c^* = \frac{c}{c_{\text{limit}}},\ t^* = t\frac{\dot{V}}{L^3},\ A_s^* = \frac{A_s}{L^2},\ k_w^* = k_w\frac{L^2}{\dot{V}},\ \text{and}\ S^* = \frac{S}{c_{\text{limit}}\dot{V}}$$

($b$) We substitute these into the equation to generate the nondimensionalized equation,

$$V^*L^3\frac{d\left(c^*c_{\text{limit}}\right)}{d\left(t^*\frac{L^3}{\dot{V}}\right)} = S^*c_{\text{limit}}\dot{V} - \dot{V}c^*c_{\text{limit}} - \left(c^*c_{\text{limit}}\right)\left(A_s^*L^2\right)\left(k_w^*\frac{\dot{V}}{L^2}\right) \qquad (1)$$

We notice that every term in Eq. 1 contains the quantity $\dot{V}c_{\text{limit}}$. We divide every term by this quantity to get a nondimensionalized form of the equation,

*Nondimensionalized equation*:

$$\boxed{V^*\frac{dc^*}{dt^*} = S^* - c^* - c^*A_s^*k_w^*}$$

**No dimensionless groups have arisen in this nondimensionalization**.

***Discussion*** Since all the characteristic scales disappear, no dimensionless groups have arisen. Since there are no dimensionless parameters, one solution in nondimensionalized variables is valid for all combinations of $L$, $\dot{V}$, and $c_{\text{limit}}$.

**7-29**

**Solution** We are to nondimensionalize the equation, and identify the dimensionless parameters that appear in the nondimensionalized equation.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible.

***Analysis*** We plug the nondimensionalized variables into the equation. For example, $u = u^*U$ and $x = x^*L$ in the first term. The result is

$$\frac{U}{L}\frac{\partial u^*}{\partial x^*}+\frac{U}{L}\frac{\partial v^*}{\partial y^*}+\frac{U}{L}\frac{\partial w^*}{\partial z^*}=0$$

or, after simplifying,

*Nondimensionalized incompressible flow relationship*:

$$\boxed{\frac{\partial u^*}{\partial x^*}+\frac{\partial v^*}{\partial y^*}+\frac{\partial w^*}{\partial z^*}=0} \qquad (1)$$

**There are no nondimensional parameters in the nondimensionalized equation.** The original equation comes from pure kinematics – there are no fluid properties involved in the equation, and therefore it is not surprising that no nondimensional parameters appear in the nondimensionalized form of the equation, Eq. 1.

***Discussion*** We show in Chap. 9 that the equation given in this problem is the differential equation for conservation of mass for an incompressible flow field – the *incompressible continuity equation*.

---

**7-30**

**Solution** We are to nondimensionalize the equation of motion and identify the dimensionless parameters that appear in the nondimensionalized equation.

***Analysis*** We plug in the nondimensionalized variables. For example, $u = u^*V$ and $x = x^*L$ in the first term. The result is

$$\frac{L^3 f}{L^3}\frac{1}{V^*}\frac{DV^*}{Dt^*}=\frac{V}{L}\frac{\partial u^*}{\partial x^*}+\frac{V}{L}\frac{\partial v^*}{\partial y^*}+\frac{V}{L}\frac{\partial w^*}{\partial z^*}$$

or, after simplifying,

$$\left(\frac{fL}{V}\right)\frac{1}{V^*}\frac{DV^*}{Dt^*}=\frac{\partial u^*}{\partial x^*}+\frac{\partial v^*}{\partial y^*}+\frac{\partial w^*}{\partial z^*} \qquad (1)$$

We recognize the nondimensional parameter in parentheses in Eq. 1 as St, the **Strouhal number**. We can re-write Eq. 1 as

*Nondimensionalized oscillating compressible flow relationship*:

$$\boxed{\mathrm{St}\frac{1}{V^*}\frac{DV^*}{Dt^*}=\frac{\partial u^*}{\partial x^*}+\frac{\partial v^*}{\partial y^*}+\frac{\partial w^*}{\partial z^*}}$$

***Discussion*** We show in Chap. 9 that the given equation of motion is the differential equation for conservation of mass for an unsteady, compressible flow field – the *general continuity equation*. We may also use angular frequency $\omega$ (radians per second) in place of physical frequency $f$ (cycles per second), with the same result.

---

**7-31**

**Solution** We are to determine the primary dimensions of the stream function, nondimensionalize the variables, and then re-write the definition of $\psi$ in nondimensionalized form.

***Assumptions*** **1** The flow is incompressible. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** (*a*) We use that fact that all equations must be dimensionally homogeneous. We solve for the dimensions of $\psi$,

*Primary dimensions of stream function*:
$$\{\psi\} = \{u\}\times\{y\} = \left\{\frac{\mathrm{L}}{\mathrm{t}}\times\mathrm{L}\right\} = \left\{\frac{\mathbf{L}^2}{\mathbf{t}}\right\}$$

Or, in exponent form, $\{\psi\}$ = $\mathbf{\{L^2\,t^{-1}\}}$.

(*b*) We nondimensionalize the variables by inspection according to their dimensions,

*Nondimensionalized variables*:
$$x^* = \frac{x}{L} \qquad y^* = \frac{y}{L} \qquad u^* = u\frac{t}{L} \qquad v^* = v\frac{t}{L} \qquad \psi^* = \psi\frac{t}{L^2}$$

(*c*) We generate the nondimensionalized equations,

$$u^*\left(\frac{L}{t}\right) = \frac{\partial\psi^*\left(\frac{L^2}{t}\right)}{\partial y^*\left(L\right)} \qquad v^*\left(\frac{L}{t}\right) = -\frac{\partial\psi^*\left(\frac{L^2}{t}\right)}{\partial x^*\left(L\right)}$$

We notice that every term in both parts of the above equation contains the ratio *L*/*t*. We divide every term by *L*/*t* to get the final nondimensionalized form of the equations,

*Nondimensionalized stream function equations*:
$$u^* = \frac{\partial\psi^*}{\partial y^*} \qquad v^* = -\frac{\partial\psi^*}{\partial x^*}$$

**No dimensionless groups have arisen in this nondimensionalization**.

***Discussion*** Since all the nondimensionalized variables scale with *L* and *t*, no dimensionless groups have arisen.

**7-32**

**Solution** We are to nondimensionalize the equation of motion and identify the dimensionless parameters that appear in the nondimensionalized equation.

***Analysis*** We plug the nondimensionalized variables into the equation. For example, $t = t^*/\omega$ and $\vec{V} = V_\infty \vec{V}^*$ in the first term on the right hand side. The result is

$$\omega^2 L\left(\vec{F}/m\right)^* = \omega U \frac{\partial \vec{V}^*}{\partial t^*} + \frac{U^2}{L}\left(\vec{V}^* \cdot \vec{\nabla}^*\right)\vec{V}^*$$

or, after simplifying by multiplying each term by $L/V_\infty^2$,

$$\left(\frac{\omega L}{V_\infty}\right)^2 \left(\vec{F}/m\right)^* = \left(\frac{\omega L}{V_\infty}\right)\frac{\partial \vec{V}^*}{\partial t^*} + \left(\vec{V}^* \cdot \vec{\nabla}^*\right)\vec{V}^* \qquad (1)$$

We recognize the nondimensional parameter in parentheses in Eq. 1 as St, the **Strouhal number**. We re-write Eq. 1 as

*Nondimensionalized Newton's second law for incompressible oscillatory flow*:

$$\left(\text{St}\right)^2 \left(\vec{F}/m\right)^* = \left(\text{St}\right)\frac{\partial \vec{V}^*}{\partial t^*} + \left(\vec{V}^* \cdot \vec{\nabla}^*\right)\vec{V}^*$$

***Discussion*** We used angular frequency $\omega$ in this problem. The same result would be obtained if we used physical frequency. Equation 1 is the basis for forming the differential equation for conservation of linear momentum for an unsteady, incompressible flow field.

---

**7-33**

**Solution** We are to nondimensionalize the Bernoulli equation and generate an expression for the pressure coefficient.

***Assumptions*** **1** The flow is incompressible. **2** Gravitational terms in the Bernoulli equation are negligible compared to the other terms.

***Analysis*** We nondimensionalize the equation by dividing each term by the **dynamic pressure**, $\frac{1}{2}\rho V_\infty^2$,

*Nondimensionalization*:

$$\frac{P}{\frac{1}{2}\rho V_\infty^2} + \frac{V^2}{V_\infty^2} = \frac{P_\infty}{\frac{1}{2}\rho V_\infty^2} + 1$$

Rearranging,

*Pressure coefficient*:

$$C_p = \frac{P - P_\infty}{\frac{1}{2}\rho V_\infty^2} = 1 - \frac{V^2}{V_\infty^2}$$

***Discussion*** Pressure coefficient is a useful dimensionless parameter that is inversely related to local air speed – as local air speed $V$ increases, $C_p$ decreases.

---

## Dimensional Analysis and Similarity

### 7-34C

**Solution** We are to list the three primary purposes of dimensional analysis.

***Analysis*** The three primary purposes of *dimensional analysis* are:

1. **To generate nondimensional parameters that help in the design of experiments and in the reporting of experimental results**.
2. **To obtain scaling laws so that prototype performance can be predicted from model performance**.
3. **To (sometimes) predict trends in the relationship between parameters**.

***Discussion*** Dimensional analysis is most useful for difficult problems that cannot be solved analytically.

### 7-35C

**Solution** We are to list and describe the three necessary conditions for complete similarity between a model and a prototype.

***Analysis*** The three necessary conditions for complete similarity between a model and a prototype are:

1. **Geometric similarity –** the model must be the **same shape** as the prototype, but scaled by some constant scale factor.
2. **Kinematic similarity – the velocity at any point in the model flow must be proportional** (by a constant scale factor) to the velocity at the corresponding point in the prototype flow.
3. **Dynamic similarity – all *forces* in the model flow scale by a constant factor to corresponding forces in the prototype flow**.

***Discussion*** Complete similarity is achievable only when all three of the above similarity conditions are met.

### 7-36

**Solution** For a scale model of a submarine being tested in air, we are to calculate the wind tunnel speed required to achieve similarity with the prototype submarine that moves through water at a given speed.

***Assumptions*** **1** Compressibility of the air is assumed to be negligible. **2** The wind tunnel walls are far enough away so as to not interfere with the aerodynamic drag on the model sub. **3** The model is geometrically similar to the prototype.

***Properties*** For water at $T$ = 15°C and atmospheric pressure, $\rho$ = 999.1 kg/m$^3$ and $\mu$ = 1.138 × 10$^{-3}$ kg/m·s. For air at $T$ = 25°C and atmospheric pressure, $\rho$ = 1.184 kg/m$^3$ and $\mu$ = 1.849 × 10$^{-5}$ kg/m·s.

***Analysis*** Similarity is achieved when the Reynolds number of the model is equal to that of the prototype,

*Similarity*:
$$\mathrm{Re_m} = \frac{\rho_\mathrm{m} V_\mathrm{m} L_\mathrm{m}}{\mu_\mathrm{m}} = \mathrm{Re_p} = \frac{\rho_\mathrm{p} V_\mathrm{p} L_\mathrm{p}}{\mu_\mathrm{p}} \qquad (1)$$

We solve Eq. 1 for the unknown wind tunnel speed,

$$V_\mathrm{m} = V_\mathrm{p}\left(\frac{\mu_\mathrm{m}}{\mu_\mathrm{p}}\right)\left(\frac{\rho_\mathrm{p}}{\rho_\mathrm{m}}\right)\left(\frac{L_\mathrm{p}}{L_\mathrm{m}}\right)$$
$$= (0.560\ \mathrm{m/s})\left(\frac{1.849\times10^{-5}\ \mathrm{kg/m\cdot s}}{1.138\times10^{-3}\ \mathrm{kg/m\cdot s}}\right)\left(\frac{999.1\ \mathrm{kg/m^3}}{1.184\ \mathrm{kg/m^3}}\right)(8) = \mathbf{61.4\ m/s}$$

***Discussion*** At this air temperature, the speed of sound is around 346 m/s. Thus the Mach number in the wind tunnel is equal to 61.4/346 = 0.177. This is sufficiently low that the incompressible flow approximation is reasonable.

**7-37**

**Solution** For a scale model of a submarine being tested in air, we are to calculate the wind tunnel speed required to achieve similarity with the prototype submarine that moves through water at a given speed.

***Assumptions*** **1** Compressibility of the air is assumed to be negligible. **2** The wind tunnel walls are far enough away so as to not interfere with the aerodynamic drag on the model sub. **3** The model is geometrically similar to the prototype.

***Properties*** For water at $T$ = 15°C and atmospheric pressure, $\rho$ = 999.1 kg/m³ and $\mu$ = 1.138 × 10⁻³ kg/m·s. For air at $T$ = 25°C and atmospheric pressure, $\rho$ = 1.184 kg/m³ and $\mu$ = 1.849 × 10⁻⁵ kg/m·s.

***Analysis*** Similarity is achieved when the Reynolds number of the model is equal to that of the prototype,

*Similarity*:

$$\text{Re}_\text{m} = \frac{\rho_\text{m} V_\text{m} L_\text{m}}{\mu_\text{m}} = \text{Re}_\text{p} = \frac{\rho_\text{p} V_\text{p} L_\text{p}}{\mu_\text{p}} \qquad (1)$$

We solve Eq. 1 for the unknown wind tunnel speed,

$$V_\text{m} = V_\text{p}\left(\frac{\mu_\text{m}}{\mu_\text{p}}\right)\left(\frac{\rho_\text{p}}{\rho_\text{m}}\right)\left(\frac{L_\text{p}}{L_\text{m}}\right) = 0.560 \text{ m/s}\left(\frac{1.849\times10^{-5}\text{ kg/m}\cdot\text{s}}{1.138\times10^{-3}\text{ kg/m}\cdot\text{s}}\right)\left(\frac{999.1\text{ kg/m}^3}{1.184\text{ kg/m}^3}\right)(24) = \mathbf{184\text{ m/s}}$$

At this air temperature, the speed of sound is around 346 m/s. Thus the Mach number in the wind tunnel is equal to 184/346 = 0.532. **The Mach number is sufficiently high that the incompressible flow approximation is not reasonable**. The wind tunnel should be run at a flow speed at which the Mach number is less than one-third of the speed of sound. At this lower speed, the Reynolds number of the model will be too small, but the results may still be usable, either by extrapolation to higher Re, or if we are fortunate enough to have Reynolds number independence, as discussed in Section 7-5.

***Discussion*** It is also unlikely that a small instructional wind tunnel can achieve such a high speed.

## 7-38E

**Solution** For a prototype parachute and its model we are to calculate drag coefficient, and determine the wind tunnel speed that ensures dynamic similarity. Then we are to estimate the aerodynamic drag on the model.

***Assumptions*** **1** The model is geometrically similar to the prototype.

***Properties*** For air at 60°F and standard atmospheric pressure, $\rho$ = 0.07633 lbm/ft$^3$ and $\mu$ = 1.213 × 10$^{-5}$ lbm/ft·s.

***Analysis*** (*a*) The aerodynamic drag on the prototype parachute is equal to the total weight. We can then easily calculate the drag coefficient $C_D$,

*Drag coefficient*:
$$C_D = \frac{F_{D,\mathrm{p}}}{\frac{1}{2}\rho_\mathrm{p} V_\mathrm{p}^{\,2} A_\mathrm{p}} = \frac{230 \text{ lbf}}{\frac{1}{2}\left(0.07633 \text{ lbm/ft}^3\right)\left(20 \text{ ft/s}\right)^2 \pi \frac{\left(24 \text{ ft}\right)^2}{4}}\left(\frac{32.2 \text{ lbm ft}}{\text{lbf s}^2}\right) = \mathbf{1.07}$$

(*b*) We must match model and prototype Reynolds numbers in order to achieve dynamic similarity,

*Similarity*:
$$\mathrm{Re}_\mathrm{m} = \frac{\rho_\mathrm{m} V_\mathrm{m} L_\mathrm{m}}{\mu_\mathrm{m}} = \mathrm{Re}_\mathrm{p} = \frac{\rho_\mathrm{p} V_\mathrm{p} L_\mathrm{p}}{\mu_\mathrm{p}} \quad (1)$$

We solve Eq. 1 for the unknown wind tunnel speed,

*Wind tunnel speed*:
$$V_\mathrm{m} = V_\mathrm{p}\left(\frac{\mu_\mathrm{m}}{\mu_\mathrm{p}}\right)\left(\frac{\rho_\mathrm{p}}{\rho_\mathrm{m}}\right)\left(\frac{L_\mathrm{p}}{L_\mathrm{m}}\right) = \left(20 \text{ ft/s}\right)\left(1\right)\left(1\right)\left(12\right) = \mathbf{240\ ft/s} \quad (2)$$

(*c*) As discussed in the text, if the fluid is the same and dynamic similarity between the model and the prototype is achieved, the aerodynamic drag force on the model is the same as that on the prototype. Thus,

*Aerodynamic drag on model*:
$$F_{D,\mathrm{m}} = F_{D,\mathrm{p}} = \mathbf{230\ lbf} \quad (3)$$

***Discussion*** We should check that the wind tunnel speed of Eq. 2 is not too high that the incompressibility approximation becomes invalid. The Mach number at this speed is about 240/1120 = 0.214. Since this is less than 0.3, compressibility is not an issue in this model test. The drag force on the model is quite large, and a fairly hefty drag balance must be available to measure such a large force.

## 7-39

**Solution** We are to discuss why one would pressurize a wind tunnel.

***Analysis*** As we see in some of the example problems and homework problems in this chapter, it is often difficult to achieve a high-enough wind tunnel speed to match the Reynolds number between a small model and a large prototype. Even if we were able to match the speed, the Mach number would often be too high. A pressurized wind tunnel has higher density air. At the same Reynolds number, the larger density leads to a lower air speed requirement. In other words, **a pressurized wind tunnel can achieve higher Reynolds numbers for the same scale model**.

If the pressure were to be increased by a factor of 1.5, the air density would also go up by a factor of 1.5 (ideal gas law), assuming that the air temperature remains constant. Then the Reynolds number, Re = $\rho VL/\mu$, would go up by approximately **1.5**. Note that we are also assuming that the viscosity does not change significantly with pressure, which is a reasonable assumption.

***Discussion*** The speed of sound is not a strong function of pressure, so Mach number is not affected significantly by pressurizing the wind tunnel. However, the *power* requirement for the wind tunnel blower increases significantly as air density is increased, so this must be taken into account when designing the wind tunnel.

---

## 7-40

**Solution** We are to estimate the drag on a prototype submarine in water, based on aerodynamic drag measurements performed in a wind tunnel.

***Assumptions*** **1** The model is geometrically similar. **2** The wind tunnel is run at conditions which ensure similarity between model and prototype.

***Properties*** For water at $T$ = 15°C and atmospheric pressure, $\rho$ = 999.1 kg/m$^3$ and $\mu$ = 1.138 × 10$^{-3}$ kg/m·s. For air at $T$ = 25°C and atmospheric pressure, $\rho$ = 1.184 kg/m$^3$ and $\mu$ = 1.849 × 10$^{-5}$ kg/m·s.

***Analysis*** Since the Reynolds numbers have been matched, the nondimensionalized drag coefficient of the model equals that of the prototype,

$$\frac{F_{D,\mathrm{m}}}{\rho_\mathrm{m} V_\mathrm{m}^{\,2} L_\mathrm{m}^{\,2}} = \frac{F_{D,\mathrm{p}}}{\rho_\mathrm{p} V_\mathrm{p}^{\,2} L_\mathrm{p}^{\,2}} \qquad (1)$$

We solve Eq. 1 for the unknown aerodynamic drag force on the prototype, $F_{D,\mathrm{p}}$,

$$F_{D,\mathrm{p}} = F_{D,\mathrm{m}} \left(\frac{\rho_\mathrm{p}}{\rho_\mathrm{m}}\right)\left(\frac{V_\mathrm{p}}{V_\mathrm{m}}\right)^2 \left(\frac{L_\mathrm{p}}{L_\mathrm{m}}\right)^2 = (2.3\ \mathrm{N})\left(\frac{999.1\ \mathrm{kg/m\cdot s}}{1.184\ \mathrm{kg/m\cdot s}}\right)\left(\frac{0.560\ \mathrm{m/s}}{61.4\ \mathrm{m/s}}\right)^2 (8)^2 = \mathbf{10.3\ N}$$

where we have used the wind tunnel speed calculated in Problem 7-36.

***Discussion*** Although the prototype moves at a much slower speed than the model, the density of water is much higher than that of air, and the prototype is eight times larger than the model. When all of these factors are combined, the drag force on the prototype is much larger than that on the model.

---

## 7-41E

**Solution** The concept of similarity will be utilized to determine the speed of the wind tunnel.

***Assumptions*** **1** Compressibility of the air is ignored (the validity of this assumption will be discussed later). **2** The wind tunnel walls are far enough away so as to not interfere with the aerodynamic drag on the model car. **3** The model is geometrically similar to the prototype. **4** Both the air in the wind tunnel and the air flowing over the prototype car are at standard atmospheric pressure.

***Properties*** For air at $T = 25^\circ$C and atmospheric pressure, $\rho = 1.184$ kg/m$^3$ and $\mu = 1.849 \times 10^{-5}$ kg/m·s.

***Analysis*** Since there is only one independent Π in this problem, similarity is achieved if $\Pi_{2,m} = \Pi_{2,p}$, where $\Pi_2$ is the Reynolds number. Thus, we can write

$$\Pi_{2,\mathrm{m}} = \mathrm{Re}_\mathrm{m} = \frac{\rho_\mathrm{m} V_\mathrm{m} L_\mathrm{m}}{\mu_\mathrm{m}} = \Pi_{2,\mathrm{p}} = \mathrm{Re}_\mathrm{p} = \frac{\rho_\mathrm{p} V_\mathrm{p} L_\mathrm{p}}{\mu_\mathrm{p}}$$

which can be solved for the unknown wind tunnel speed for the model tests, $V_m$,

$$V_\mathrm{m} = V_\mathrm{p}\left(\frac{\mu_\mathrm{m}}{\mu_\mathrm{p}}\right)\left(\frac{\rho_\mathrm{p}}{\rho_\mathrm{m}}\right)\left(\frac{L_\mathrm{p}}{L_\mathrm{m}}\right) = (60.0 \text{ mph})(1)(1)(4) = \mathbf{240\ mph}$$

Thus, to ensure similarity, the wind tunnel should be run at 240 miles per hour (to three significant digits).

***Discussion*** This speed is quite high, and the wind tunnel may not be able to run at that speed. We were never given the actual length of either car, but the ratio of $L_p$ to $L_m$ is known because the prototype is four times larger than the scale model. The problem statement contains a mixture of SI and English units, but it does not matter since we use ratios in the algebra.

---

## 7-42E

**Solution** We are to estimate the drag on a prototype car, based on aerodynamic drag measurements performed in a wind tunnel.

***Assumptions*** **1** The model is geometrically similar. **2** The wind tunnel is run at conditions which ensure similarity between model and prototype.

***Properties*** For air at $T = 25^\circ$C and atmospheric pressure, $\rho = 1.184$ kg/m$^3$ and $\mu = 1.849 \times 10^{-5}$ kg/m·s.

***Analysis*** Following the example in the text, since the Reynolds numbers have been matched, the nondimensionalized drag coefficient of the model equals that of the prototype,

$$\frac{F_{D,\mathrm{m}}}{\rho_\mathrm{m} V_\mathrm{m}^2 L_\mathrm{m}^2} = \frac{F_{D,\mathrm{p}}}{\rho_\mathrm{p} V_\mathrm{p}^2 L_\mathrm{p}^2} \qquad (1)$$

We solve Eq. 1 for the unknown aerodynamic drag force on the prototype, $F_{D,p}$,

$$F_{D,\mathrm{p}} = F_{D,\mathrm{m}}\left(\frac{\rho_\mathrm{p}}{\rho_\mathrm{m}}\right)\left(\frac{V_\mathrm{p}}{V_\mathrm{m}}\right)^2\left(\frac{L_\mathrm{p}}{L_\mathrm{m}}\right)^2 = (36.5 \text{ lbf})(1)\left(\frac{60.0 \text{ mph}}{240 \text{ mph}}\right)^2(4)^2 = \mathbf{36.5\ lbf}$$

where we have used the wind tunnel speed calculated in the previous problem.

***Discussion*** Since the air properties of the wind tunnel are identical to those of the air flowing around the prototype car, it turns out that the aerodynamic drag force on the prototype is the same as that on the model. This would not be the case if the wind tunnel air were at a different temperature or pressure compared to that of the prototype.

---

**7-43**

**Solution** We are to discuss whether cold or hot air in a wind tunnel is better, and we are to support our answer by comparing air at two given temperatures.

***Properties*** For air at atmospheric pressure and at $T = 10^{o}$C, $\rho = 1.246$ kg/m$^3$ and $\mu = 1.778 \times 10^{-5}$ kg/m·s. At $T = 50^{o}$C, $\rho = 1.092$ kg/m$^3$ and $\mu = 1.963 \times 10^{-5}$ kg/m·s.

***Analysis*** As we see in some of the example problems and homework problems in this chapter, it is often difficult to achieve a high-enough wind tunnel speed to match the Reynolds number between a small model and a large prototype. Even if we were able to match the speed, the Mach number would often be too high. Cold air has higher density than warm air. In addition, the viscosity of cold air is lower than that of hot air. Thus, at the same Reynolds number, the colder air leads to a lower air speed requirement. In other words, **a cold wind tunnel can achieve higher Reynolds numbers than can a hot wind tunnel for the same scale model, all else being equal**. We support our conclusion by comparing air at two temperatures,

*Comparison of Reynolds numbers*:

$$\frac{\text{Re}_{\text{cold}}}{\text{Re}_{\text{hot}}} = \frac{\dfrac{\rho_{\text{cold}} VL}{\mu_{\text{cold}}}}{\dfrac{\rho_{\text{hot}} VL}{\mu_{\text{hot}}}} = \frac{\rho_{\text{cold}}}{\rho_{\text{hot}}} \frac{\mu_{\text{hot}}}{\mu_{\text{cold}}} = \frac{1.246 \text{ kg/m}^3}{1.092 \text{ kg/m}^3} \frac{1.963\times10^{-5} \text{ kg/m}\cdot\text{s}}{1.778\times10^{-5} \text{ kg/m}\cdot\text{s}} = \mathbf{1.26}$$

Thus we see that **the colder wind tunnel can achieve approximately 26% higher Reynolds number, all else being equal**.

***Discussion*** There are other issues however. First of all, the denser air of the cold wind tunnel is harder to pump – the cold wind tunnel may not be able to achieve the same wind speed as the hot wind tunnel. Furthermore, the speed of sound is proportional to the square root of temperature. Thus, at colder temperatures, the Mach number is higher than at warmer temperatures for the same value of $V$, and compressibility effects are therefore more significant at lower temperatures.

---

**7-44**

**Solution** We are to calculate the speed and angular velocity (rpm) of a spinning baseball in a water channel such that flow conditions are dynamically similar to that of the actual baseball moving and spinning in air.

***Properties*** For air at $T = 20^{o}$C and atmospheric pressure, $\rho = 1.204$ kg/m$^3$ and $\mu = 1.825 \times 10^{-5}$ kg/m·s. For water at $T = 20^{o}$C and atmospheric pressure, $\rho = 998.0$ kg/m$^3$ and $\mu = 1.002 \times 10^{-3}$ kg/m·s.

***Analysis*** The model (in the water) and the prototype (in the air) are actually the same baseball, so their characteristic lengths are equal, $L_{\text{m}} = L_{\text{p}}$. We match Reynolds number,

$$\text{Re}_{\text{m}} = \frac{\rho_{\text{m}} V_{\text{m}} L_{\text{m}}}{\mu_{\text{m}}} = \text{Re}_{\text{p}} = \frac{\rho_{\text{p}} V_{\text{p}} L_{\text{p}}}{\mu_{\text{p}}} \qquad (1)$$

and solve for the required water tunnel speed for the model tests, $V_{\text{m}}$,

$$V_{\text{m}} = V_{\text{p}} \left(\frac{\mu_{\text{m}}}{\mu_{\text{p}}}\right)\left(\frac{\rho_{\text{p}}}{\rho_{\text{m}}}\right)\left(\frac{L_{\text{p}}}{L_{\text{m}}}\right) = (80.0 \text{ mph})\left(\frac{1.002\times10^{-3} \text{ kg/m}\cdot\text{s}}{1.825\times10^{-5} \text{ kg/m}\cdot\text{s}}\right)\left(\frac{1.204 \text{ kg/m}^3}{998.0 \text{ kg/m}^3}\right)(1) = \mathbf{5.30 \text{ mph}} \qquad (2)$$

We also match Strouhal numbers, recognizing that $\dot{n}$ is proportional to $f$,

$$\text{St}_{\text{m}} = \frac{f_{\text{m}} L_{\text{m}}}{V_{\text{m}}} = \text{St}_{\text{p}} = \frac{f_{\text{p}} L_{\text{p}}}{V_{\text{p}}} \quad \rightarrow \quad \frac{\dot{n}_{\text{m}} L_{\text{m}}}{V_{\text{m}}} = \frac{\dot{n}_{\text{p}} L_{\text{p}}}{V_{\text{p}}} \qquad (3)$$

from which we solve for the required spin rate in the water tunnel,

$$\dot{n}_{\text{m}} = \dot{n}_{\text{p}} \left(\frac{L_{\text{p}}}{L_{\text{m}}}\right)\left(\frac{V_{\text{m}}}{V_{\text{p}}}\right) = (300 \text{ rpm})(1)\left(\frac{5.30 \text{ mph}}{80.0 \text{ mph}}\right) = \mathbf{19.9 \text{ rpm}} \qquad (4)$$

***Discussion*** Because of the difference in fluid properties between air and water, the required water tunnel speed is much lower than that in air. In addition, the spin rate is much lower, making flow visualization easier.

---

## Dimensionless Parameters and the Method of Repeating Variables

### 7-45

**Solution** We are to verify that the Archimedes number is dimensionless.

***Analysis*** Archimedes number is defined as

*Archimedes number*:
$$\text{Ar} = \frac{\rho_s g L^3}{\mu^2}\left(\rho_s - \rho\right) \qquad (1)$$

We know the primary dimensions of density, gravitational acceleration, length, and viscosity. Thus,

*Primary dimensions of Archimedes number*:
$$\{\text{Ar}\} = \left\{ \frac{\frac{\text{m}}{\text{L}^3}\frac{\text{L}}{\text{t}^2}\text{L}^3}{\frac{\text{m}^2}{\text{L}^2\text{t}^2}} \frac{\text{m}}{\text{L}^3} \right\} = \{\mathbf{1}\} \qquad (2)$$

***Discussion*** If the primary dimensions were *not* unity, we would assume that we made an error in the dimensions of one or more of the parameters.

### 7-46

**Solution** We are to verify that the Grashof number is dimensionless.

***Analysis*** Grashof number is defined as

*Grashof number*:
$$\text{Gr} = \frac{g\beta\left|\Delta T\right| L^3 \rho^2}{\mu^2} \qquad (1)$$

We know the primary dimensions of density, gravitational acceleration, length, temperature, and viscosity. The dimensions of coefficient of thermal expansion $\beta$ are 1/temperature. Thus,

*Primary dimensions of Grashof number*:
$$\{\text{Gr}\} = \left\{ \frac{\frac{\text{L}}{\text{t}^2}\frac{1}{\text{T}}\text{TL}^3\frac{\text{m}^2}{\text{L}^6}}{\frac{\text{m}^2}{\text{L}^2\text{t}^2}} \right\} = \{\mathbf{1}\} \qquad (2)$$

***Discussion*** If the primary dimensions were *not* unity, we would assume that we made an error in the dimensions of one or more of the parameters.

**7-47**

**Solution** We are to verify that the Rayleigh number is dimensionless, and determine what other established nondimensional parameter is formed by the ratio of Ra and Gr.

***Analysis*** Rayleigh number is defined as

*Rayleigh number*:
$$\text{Ra} = \frac{g\beta\left|\Delta T\right|L^3\rho^2 c_p}{k\mu} \qquad (1)$$

We know the primary dimensions of density, gravitational acceleration, length, temperature, and viscosity. The dimensions of coefficient of thermal expansion $\beta$ are 1/temperature, those of specific heat $c_p$ are length$^2$/time$^2$·temperature (Problem 7-14), and those of thermal conductivity $k$ are mass·length/time$^3$·temperature. Thus,

*Primary dimensions of Rayleigh number*:
$$\{\text{Ra}\} = \left\{\frac{\frac{\text{L}}{\text{t}^2}\frac{1}{\text{T}}\text{TL}^3\frac{\text{m}^2}{\text{L}^6}\frac{\text{L}^2}{\text{t}^2\text{T}}}{\frac{\text{mL}}{\text{t}^3\text{T}}\frac{\text{m}}{\text{Lt}}}\right\} = \{\mathbf{1}\} \qquad (2)$$

We take the ratio of Ra and Gr:

*Ratio of Rayleigh number and Grashof number*:
$$\frac{\text{Ra}}{\text{Gr}} = \frac{\dfrac{g\beta\left|\Delta T\right|L^3\rho^2 c_p}{k\mu}}{\dfrac{g\beta\left|\Delta T\right|L^3\rho^2}{\mu^2}} = \frac{c_p\mu}{k} \qquad (3)$$

We recognize Eq. 3 as the **Prandtl number**,

*Prandtl number*:
$$\boxed{\text{Pr} = \frac{\text{Ra}}{\text{Gr}} = \frac{c_p\mu}{k} = \frac{\rho c_p\mu}{\rho k} = \frac{\nu}{\alpha}} \qquad (4)$$

***Discussion*** Many of the established nondimensional parameters are formed by the ratio or product of two (or more) other established nondimensional parameters.

**7-48**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are five parameters in this problem; $n = 5$,

*List of relevant parameters*: $$h = f\left(\omega, \rho, g, R\right) \qquad n = 5 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $h$ | $\omega$ | $\rho$ | $g$ | $R$ |
|---|---|---|---|---|
| $\{L^1\}$ | $\{t^{-1}\}$ | $\{m^1L^{-3}\}$ | $\{L^1t^{-2}\}$ | $\{L^1\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 5 - 3 = 2$$

**Step 4** We need to choose three repeating parameters since $j = 3$. Following the guidelines outlined in this chapter, we elect not to pick the viscosity. We choose

*Repeating parameters*: $\omega$, $\rho$, and $R$

**Step 5** The dependent Π is generated:

$$\Pi_1 = h\omega^{a_1}\rho^{b_1}R^{c_1} \qquad \{\Pi_1\} = \left\{\left(L^1\right)\left(t^{-1}\right)^{a_1}\left(m^1L^{-3}\right)^{b_1}\left(L^1\right)^{c_1}\right\}$$

*mass*: $\{m^0\} = \{m^{b_1}\}$ $\qquad 0 = b_1 \qquad b_1 = 0$

*time*: $\{t^0\} = \{t^{-a_1}\}$ $\qquad 0 = -a_1 \qquad a_1 = 0$

*length*: $\{L^0\} = \{L^1L^{-3b_1}L^{c_1}\}$ $\qquad 0 = 1 - 3b_1 + c_1 \qquad c_1 = -1$

The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{h}{R}$$

The second Pi (the only independent Π in this problem) is generated:

$$\Pi_2 = g\omega^{a_2}\rho^{b_2}R^{c_2} \qquad \{\Pi_2\} = \left\{\left(L^1t^{-2}\right)\left(t^{-1}\right)^{a_2}\left(m^1L^{-3}\right)^{b_2}\left(L^1\right)^{c_2}\right\}$$

*mass:* $\{m^0\} = \{m^{b_2}\}$ $\qquad 0 = b_2 \qquad b_2 = 0$

*time:* $\{t^0\} = \{t^{-2}t^{-a_2}\}$ $\qquad 0 = -2 - a_2 \qquad a_2 = -2$

*length:* $\{L^0\} = \{L^1 L^{-3b_2} L^{c_2}\}$ $\quad 0 = 1 - 3b_2 + c_2 \quad 0 = 1 + c_2 \qquad c_2 = -1$

which yields

$\Pi_2$:
$$\Pi_2 = \frac{g}{\omega^2 R}$$

If we take $\Pi_2$ to the power –1/2 and recognize that $\omega R$ is the speed of the rim, we see that $\Pi_2$ can be modified into a **Froude number**,

*Modified* $\Pi_2$:
$$\Pi_2 = \text{Fr} = \frac{\omega R}{\sqrt{gR}}$$

**Step 6** We write the final functional relationship as

*Relationship between* $\Pi$*s*:
$$\boxed{\frac{h}{R} = f\left(\text{Fr}\right)} \qquad (2)$$

***Discussion*** In the generation of the first Π, $h$ and $R$ have the same dimensions. Thus, we could have immediately written down the result, $\Pi_1 = h/R$. Notice that density $\rho$ does not appear in the result. Thus, density is not a relevant parameter after all.

**7-49**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are seven parameters in this problem; $n = 7$,

*List of relevant parameters*: $$h = f\left(\omega, \rho, g, R, t, \mu\right) \qquad n = 7 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $h$ | $\omega$ | $\rho$ | $g$ | $R$ | $t$ | $\mu$ |
|---|---|---|---|---|---|---|
| $\{L^1\}$ | $\{t^{-1}\}$ | $\{m^1L^{-3}\}$ | $\{L^1t^{-2}\}$ | $\{L^1\}$ | $\{t^1\}$ | $\{m^1L^{-1}t^{-1}\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected Πs*: $$k = n - j = 7 - 3 = 4$$

**Step 4** We need to choose three repeating parameters since $j = 3$. For convenience we choose the same repeating parameters that we used in the previous problem,

*Repeating parameters*: $\omega$, $\rho$, and $R$

**Step 5** The first two Πs are identical to those of the previous problem:

$\Pi_1$: $$\Pi_1 = \frac{h}{R}$$

and

$\Pi_2$: $$\Pi_2 = \frac{\omega R}{\sqrt{gR}}$$

where $\Pi_2$ is identified as a form of the **Froude number**. The third Π is formed with time $t$. Since repeating parameter $\omega$ has dimensions of 1/time, it is the only one that remains in the Π. Thus, without the formal algebra,

$\Pi_3$: $$\Pi_3 = \omega t$$

Finally, $\Pi_4$ is generated with liquid viscosity,

$$\Pi_4 = \mu\omega^{a_4}\rho^{b_4}R^{c_4} \qquad \{\Pi_4\} = \left\{\left(m^1L^{-1}t^{-1}\right)\left(t^{-1}\right)^{a_4}\left(m^1L^{-3}\right)^{b_4}\left(L^1\right)^{c_4}\right\}$$

*mass*: $$\{m^0\} = \{m^1m^{b_4}\} \qquad 0 = 1 + b_4 \qquad b_4 = -1$$

*time*: $$\{t^0\} = \{t^{-1}t^{-a_4}\} \qquad 0 = -1 - a_4 \qquad a_4 = -1$$

*length*: $$\{L^0\} = \{L^{-1}L^{-3b_4}L^{c_4}\} \qquad 0 = -1 - 3b_4 + c_4 \qquad c_4 = 1 + 3b_4 \qquad c_4 = -2$$

The final Π is thus

$\Pi_4$: $$\Pi_4 = \frac{\mu}{\rho\Omega R^2} \qquad (2)$$

If we invert $\Pi_4$ and recognize that $\omega R$ is the speed of the rim, it becomes clear that $\Pi_4$ of Eq. 2 can be modified into a **Reynolds number**,

*Modified* $\Pi_4$: $$\Pi_4 = \frac{\rho\omega R^2}{\mu} = \text{Re} \qquad (3)$$

**Step 6** We write the final functional relationship as

*Relationship between* $\Pi$*s*: $$\boxed{\frac{h}{R} = f\left(\text{Fr}, \omega t, \text{Re}\right)} \qquad (4)$$

***Discussion*** Notice that this time density $\rho$ *does* appear in the result. There are other acceptable answers, but this one has the most established dimensionless groups.

**7-50**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are five parameters in this problem; $n$ = 5,

*List of relevant parameters*: $$f_k = f\left(V, \rho, \mu, D\right) \qquad n = 5$$

**Step 2** The primary dimensions of each parameter are listed,

| $f_k$ | $V$ | $\rho$ | $\mu$ | $D$ |
|---|---|---|---|---|
| $\{t^{-1}\}$ | $\{L^1t^{-1}\}$ | $\{m^1L^{-3}\}$ | $\{m^1L^{-1}t^{-1}\}$ | $\{L^1\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 5 - 3 = 2$$

**Step 4** We need to choose three repeating parameters since $j$ = 3. Following the guidelines outlined in this chapter, we elect not to pick the viscosity. We choose

*Repeating parameters*: $V$, $\rho$, and $D$

**Step 5** The dependent Π is generated:

$$\Pi_1 = f_k V^{a_1} \rho^{b_1} D^{c_1} \qquad \{\Pi_1\} = \left\{\left(t^{-1}\right)\left(L^1t^{-1}\right)^{a_1}\left(m^1L^{-3}\right)^{b_1}\left(L^1\right)^{c_1}\right\}$$

*mass*: $\{m^0\} = \{m^{b_1}\}$ $\quad 0 = b_1 \quad b_1 = 0$

*time*: $\{t^0\} = \{t^{-1}t^{-a_1}\}$ $\quad 0 = -1 - a_1 \quad a_1 = -1$

*length*: $\{L^0\} = \{L^{a_1}L^{-3b_1}L^{c_1}\}$ $\quad 0 = a_1 - 3b_1 + c_1 \quad c_1 = 1$

The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{f_k D}{V} = \mathrm{St}$$

where we have identified this Pi as the **Strouhal number**.

The second Pi (the only independent Π in this problem) is generated:

$$\Pi_2 = \mu V^{a_2} \rho^{b_2} D^{c_2} \qquad \{\Pi_2\} = \left\{\left(m^1L^{-1}t^{-1}\right)\left(L^1t^{-1}\right)^{a_2}\left(m^1L^{-3}\right)^{b_2}\left(L^1\right)^{c_2}\right\}$$

*mass:* $\{m^0\} = \{m^1m^{b_2}\}$ $\quad 0 = 1 + b_2 \quad b_2 = -1$

*time:* $\{t^0\} = \{t^{-1}t^{-a_2}\}$ $\quad 0 = -1 - a_2 \quad a_2 = -1$

*length:* $\{L^0\} = \{L^{-1} L^{a_2} L^{-3b_2} L^{c_2}\}$

$$0 = -1 + a_2 - 3b_2 + c_2 \qquad c_2 = -1$$
$$0 = -1 - 1 + 3 + c_2$$

which yields

$\Pi_2$: $$\Pi_2 = \frac{\mu}{\rho V D}$$

We recognize this Π as the inverse of the **Reynolds number**. So, after inverting,

*Modified* $\Pi_2$: $$\Pi_2 = \frac{\rho V D}{\mu} = \text{Reynolds number} = \text{Re}$$

**Step 6** We write the final functional relationship as

*Relationship between* Π*s*: $$\boxed{\text{St} = f(\text{Re})}$$

***Discussion*** We cannot tell from dimensional analysis the exact form of the functional relationship. However, experiments verify that the Strouhal number is indeed a function of Reynolds number.

**7-51**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are six parameters in this problem; $n = 6$,

*List of relevant parameters*: $$f_k = f\left(V, \rho, \mu, D, c\right) \qquad n = 6 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $f_k$ | $V$ | $\rho$ | $\mu$ | $D$ | $c$ |
|---|---|---|---|---|---|
| $\{t^{-1}\}$ | $\{L^1t^{-1}\}$ | $\{m^1L^{-3}\}$ | $\{m^1L^{-1}t^{-1}\}$ | $\{L^1\}$ | $\{L^1t^{-1}\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 6 - 3 = 3$$

**Step 4** We need to choose three repeating parameters since $j = 3$. Following the guidelines outlined in this chapter, we elect not to pick the viscosity. We choose

*Repeating parameters*: $V$, $\rho$, and $D$

**Step 5** The dependent Π is generated:

$$\Pi_1 = f_k V^{a_1}\rho^{b_1}D^{c_1} \qquad \{\Pi_1\} = \left\{\left(t^{-1}\right)\left(L^1t^{-1}\right)^{a_1}\left(m^1L^{-3}\right)^{b_1}\left(L^1\right)^{c_1}\right\}$$

*mass*: $\{m^0\} = \{m^{b_1}\}$ $\qquad 0 = b_1 \qquad b_1 = 0$

*time*: $\{t^0\} = \{t^{-1}t^{-a_1}\}$ $\qquad 0 = -1 - a_1 \qquad a_1 = -1$

*length*: $\{L^0\} = \{L^{a_1}L^{-3b_1}L^{c_1}\}$ $\qquad 0 = a_1 - 3b_1 + c_1 \qquad c_1 = 1$

The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{f_k D}{V} = \text{St}$$

where we have identified this Pi as the **Strouhal number**.

The second Pi (the first independent Π in this problem) is generated:

$$\Pi_2 = \mu V^{a_2}\rho^{b_2}D^{c_2} \qquad \{\Pi_2\} = \left\{\left(m^1L^{-1}t^{-1}\right)\left(L^1t^{-1}\right)^{a_2}\left(m^1L^{-3}\right)^{b_2}\left(L^1\right)^{c_2}\right\}$$

*mass:* $\{m^0\} = \{m^1m^{b_2}\}$ $\qquad 0 = 1 + b_2 \qquad b_2 = -1$

*time:* $\{t^0\} = \{t^{-1}t^{-a_2}\}$ $\qquad 0 = -1 - a_2 \qquad a_2 = -1$

*length:* $\{L^0\} = \{L^{-1}L^{a_2}L^{-3b_2}L^{c_2}\}$ $\quad 0 = -1 + a_2 - 3b_2 + c_2$ $\quad 0 = -1 - 1 + 3 + c_2$ $\quad c_2 = -1$

which yields

$\Pi_2$: $$\Pi_2 = \frac{\mu}{\rho VD}$$

We recognize this Π as the inverse of the **Reynolds number**. So, after inverting,

*Modified* $\Pi_2$: $$\Pi_2 = \frac{\rho VD}{\mu} = \text{Reynolds number} = \text{Re}$$

The third Pi (the second independent Π in this problem) is generated:

$$\Pi_3 = cV^{a_3}\rho^{b_3}D^{c_3} \qquad \{\Pi_3\} = \left\{\left(L^1t^{-1}\right)\left(L^1t^{-1}\right)^{a_3}\left(m^1L^{-3}\right)^{b_3}\left(L^1\right)^{c_3}\right\}$$

*mass:* $\{m^0\} = \{m^{b_3}\}$ $\quad 0 = b_3$ $\quad b_3 = 0$

*time:* $\{t^0\} = \{t^{-1}t^{-a_3}\}$ $\quad 0 = -1 - a_3$ $\quad a_3 = -1$

*length:* $\{L^0\} = \{L^1L^{a_3}L^{-3b_3}L^{c_3}\}$ $\quad 0 = 1 + a_3 - 3b_3 + c_3$ $\quad 0 = 1 - 1 + c_3$ $\quad c_3 = 0$

which yields

$\Pi_3$: $$\Pi_3 = \frac{c}{V}$$

We recognize this Π as the inverse of the **Mach number**. So, after inverting,

*Modified* $\Pi_3$: $$\Pi_3 = \frac{V}{c} = \text{Mach number} = \text{Ma}$$

**Step 6** We write the final functional relationship as

*Relationship between* Π*s*: $$\boxed{\text{St} = f(\text{Re}, \text{Ma})}$$

***Discussion*** We have shown all the details. After you become comfortable with the method of repeating variables, you can do some of the algebra in your head.

**7-52**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are five parameters in this problem; $n$ = 5,

*List of relevant parameters*: $$\dot{W} = f(\omega, \rho, \mu, D) \qquad n = 5 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $\dot{W}$ | $\omega$ | $\rho$ | $\mu$ | $D$ |
|---|---|---|---|---|
| $\{m^1L^2t^{-3}\}$ | $\{t^{-1}\}$ | $\{m^1L^{-3}\}$ | $\{m^1L^{-1}t^{-1}\}$ | $\{L^1\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 5 - 3 = 2$$

**Step 4** We need to choose three repeating parameters since $j$ = 3. Following the guidelines outlined in this chapter, we elect not to pick the viscosity. We choose

*Repeating parameters*: $\omega$, $\rho$, and $D$

**Step 5** The dependent Π is generated:

$$\Pi_1 = \dot{W}\omega^{a_1}\rho^{b_1}D^{c_1} \qquad \{\Pi_1\} = \left\{(m^1L^2t^{-3})(t^{-1})^{a_1}(m^1L^{-3})^{b_1}(L^1)^{c_1}\right\}$$

*mass*: $\{m^0\} = \{m^1m^{b_1}\}$ $\qquad 0 = 1 + b_1 \qquad b_1 = -1$

*time*: $\{t^0\} = \{t^{-3}t^{-a_1}\}$ $\qquad 0 = -3 - a_1 \qquad a_1 = -3$

*length*: $\{L^0\} = \{L^2L^{-3b_1}L^{c_1}\}$ $\qquad 0 = 2 - 3b_1 + c_1 \qquad c_1 = -5$

The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{\dot{W}}{\rho D^5\omega^3} = N_P$$

where we have defined this Pi as the **power number** (Table 7-5).

The second Pi (the only independent Π in this problem) is generated:

$$\Pi_2 = \mu\omega^{a_2}\rho^{b_2}D^{c_2} \qquad \{\Pi_2\} = \left\{(m^1L^{-1}t^{-1})(t^{-1})^{a_2}(m^1L^{-3})^{b_2}(L^1)^{c_2}\right\}$$

*mass:* $\{m^0\} = \{m^1m^{b_2}\}$ $\qquad 0 = 1 + b_2 \qquad b_2 = -1$

*time:* $\{t^0\} = \{t^{-1}t^{-a_2}\}$ $0 = -1 - a_2$ $a_2 = -1$

*length:* $\{L^0\} = \{L^{-1}L^{-3b_2}L^{c_2}\}$ $0 = -1 - 3b_2 + c_2$ $c_2 = -2$

$0 = -1 + 3 + c_2$

which yields

$\Pi_2$:

$$\Pi_2 = \frac{\mu}{\rho D^2 \omega}$$

Since $D\omega$ is the speed of the tip of the rotating stirrer blade, we recognize this $\Pi$ as the inverse of a **Reynolds number**. So, after inverting,

*Modified* $\Pi_2$:

$$\Pi_2 = \frac{\rho D^2 \omega}{\mu} = \frac{\rho (D\omega) D}{\mu} = \text{Reynolds number} = \text{Re}$$

**Step 6** We write the final functional relationship as

*Relationship between* $\Pi$*s*:

$$\boxed{N_P = f(\text{Re})} \qquad (2)$$

***Discussion*** After some practice you should be able to do some of the algebra with the exponents in your head. Also, we usually expect a type of Reynolds number when we combine viscosity with a density, a length, and some kind of speed, be it angular speed or linear speed.

---

## 7-53

**Solution** We are to determine the dimensionless relationship between the given parameters

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The dimensional analysis is identical to the previous problem except that we add two additional independent parameters, both of which have dimensions of length. The two $\Pi$s of the previous problem remain. We get two additional $\Pi$s since *n* is now equal to 7 instead of 5. There is no need to go through all the algebra for the two additional $\Pi$s – since their dimensions match those of one of the repeating variables (*D*), we know that all the exponents in the $\Pi$ will be zero except the exponent for *D*, which will be –1. The two additional $\Pi$s are

$\Pi_3$ and $\Pi_4$:

$$\Pi_3 = \frac{D_{\text{tank}}}{D} \qquad \Pi_4 = \frac{h_{\text{surface}}}{D}$$

The final functional relationship is

*Relationship between* $\Pi$*s*:

$$\boxed{N_P = f\left(\text{Re}, \frac{D_{\text{tank}}}{D}, \frac{h_{\text{surface}}}{D}\right)} \qquad (1)$$

***Discussion*** We could also manipulate our $\Pi$s so that we have other length ratios like $h_{\text{surface}}/D_{\text{tank}}$, etc. Any such combination is acceptable.

**7-54**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are five parameters in this problem; $n = 5$,

*List of relevant parameters*: $$\delta = f\left(x, V, \rho, \mu\right) \qquad n = 5 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $\delta$ | $x$ | $V$ | $\rho$ | $\mu$ |
|---|---|---|---|---|
| $\{L^1\}$ | $\{L^1\}$ | $\{L^1t^{-1}\}$ | $\{m^1L^{-3}\}$ | $\{m^1L^{-1}t^{-1}\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 5 - 3 = 2$$

**Step 4** We need to choose three repeating parameters since $j$ = 3. We pick length scale $x$, density $\rho$, and freestream velocity $V$.

*Repeating parameters*: $x$, $\rho$, and $V$

**Step 5** The Πs are generated. Note that for the first Π we can do the algebra in our heads since the relationship is very simple. Namely, the dimensions of $\delta$ are identical to those of one of the repeating variables ($x$). In such a case we know that all the exponents in the Π group are zero except the one for $x$, which is –1. The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{\delta}{x}$$

The second Π is formed with viscosity,

$$\Pi_2 = \mu x^a \rho^b V^c \qquad \{\Pi_2\} = \left\{\left(m^1L^{-1}t^{-1}\right)\left(L^1\right)^a\left(m^1L^{-3}\right)^b\left(L^1t^{-1}\right)^c\right\}$$

*mass:* $\{m^0\} = \{m^1m^b\}$ $\qquad 0 = 1 + b \qquad b = -1$

*time:* $\{t^0\} = \{t^{-1}t^{-c}\}$ $\qquad 0 = -1 - c \qquad c = -1$

*length:* $\{L^0\} = \{L^{-1}L^aL^{-3b}L^c\}$ $\qquad 0 = -1 + a - 3b + c$; $\quad 0 = -1 + a + 3 - 1 \qquad a = -1$

which yields

$\Pi_2$: $$\Pi_2 = \frac{\mu}{\rho V x}$$

We recognize this Π as the inverse of the **Reynolds number**,

*Modified* $\Pi_2$ = *Reynolds number based on x*: $\Pi_2 = \text{Re}_x \dfrac{\rho V x}{\mu}$

**Step 6** We write the final functional relationship as

*Relationship between* $\Pi$*s*: $$\boxed{\frac{\delta}{x} = f\left(\text{Re}_x\right)}$$

***Discussion*** We cannot determine the *form* of the relationship by purely dimensional reasoning since there are two $\Pi$s. However, in Chap. 10 we shall see that for a laminar boundary layer, $\Pi_1$ is proportional to the square root of $\Pi_2$.

---

## 7-55

**Solution** We are to create a scale for volume flow rate and then define an appropriate Richardson number.

***Analysis*** By "back of the envelope" reasoning (or by inspection), we define a volume flow rate scale as $L^2V$. Then the Richardson number can be defined as

*Richardson number*: $$\boxed{\text{Ri} = \frac{L^5 g \Delta\rho}{\rho \dot{V}^2} = \frac{L^5 g \Delta\rho}{\rho\left(L^2 V\right)^2} = \frac{L g \Delta\rho}{\rho V^2}} \qquad (1)$$

***Discussion*** It is perhaps more clear from the form of Eq. 1 that Richardson number is a ratio of buoyancy forces to inertial forces.

---

**7-56**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are six parameters in this problem; $n = 6$,

*List of relevant parameters*: $$u = f\left(\mu, V, h, \rho, y\right) \qquad n = 6 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $u$ | $\mu$ | $V$ | $h$ | $\rho$ | $y$ |
|---|---|---|---|---|---|
| $\{L^1t^{-1}\}$ | $\{m^1L^{-1}t^{-1}\}$ | $\{L^1t^{-1}\}$ | $\{L^1\}$ | $\{m^1L^{-3}\}$ | $\{L^1\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 6 - 3 = 3$$

**Step 4** We need to choose three repeating parameters since $j = 3$. Following the guidelines outlined in this chapter, we elect not to pick the viscosity. It is better to pick a fixed length ($h$) rather than a variable length ($y$); otherwise $y$ would appear in each Pi, which would not be desirable. We choose

*Repeating parameters*: $V$, $\rho$, and $h$

**Step 5** The dependent Π is generated:

$$\Pi_1 = uV^{a_1}\rho^{b_1}h^{c_1} \qquad \{\Pi_1\} = \left\{\left(L^1t^{-1}\right)\left(L^1t^{-1}\right)^{a_1}\left(m^1L^{-3}\right)^{b_1}\left(L^1\right)^{c_1}\right\}$$

*mass*: $\{m^0\} = \{m^{b_1}\}$ $\qquad 0 = b_1 \qquad b_1 = 0$

*time*: $\{t^0\} = \{t^{-1}t^{-a_1}\}$ $\qquad 0 = -1 - a_1 \qquad a_1 = -1$

*length*: $\{L^0\} = \{L^1L^{a_1}L^{-3b_1}L^{c_1}\}$ $\qquad 0 = 1 + a_1 - 3b_1 + c_1 \qquad c_1 = 0$

The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{u}{V}$$

The second Pi (the first independent Π in this problem) is generated:

$$\Pi_2 = \mu V^{a_2}\rho^{b_2}h^{c_2} \qquad \{\Pi_2\} = \left\{\left(m^1L^{-1}t^{-1}\right)\left(L^1t^{-1}\right)^{a_2}\left(m^1L^{-3}\right)^{b_2}\left(L^1\right)^{c_2}\right\}$$

*mass:* $\{m^0\} = \{m^1m^{b_2}\}$ $\qquad 0 = 1 + b_2 \qquad b_2 = -1$

*time:* $\{t^0\} = \{t^{-1}t^{-a_2}\}$ $0 = -1 - a_2$ $a_2 = -1$

*length:* $\{L^0\} = \{L^{-1}L^{a_2}L^{-3b_2}L^{c_2}\}$ $0 = -1 + a_2 - 3b_2 + c_2$ $c_2 = -1$
$0 = -1 - 1 + 3 + c_2$

which yields

$\Pi_2$: $$\Pi_2 = \frac{\mu}{\rho V h}$$

We recognize this Π as the inverse of the **Reynolds number**. So, after inverting,

*Modified* $\Pi_2$: $$\Pi_2 = \frac{\rho V h}{\mu} = \text{Reynolds number} = \text{Re}$$

The third Pi (the second independent Π in this problem) is generated:

$$\Pi_3 = yV^{a_3}\rho^{b_3}h^{c_3} \qquad \{\Pi_3\} = \left\{(L^1)(L^1t^{-1})^{a_3}(m^1L^{-3})^{b_3}(L^1)^{c_3}\right\}$$

*mass:* $\{m^0\} = \{m^{b_3}\}$ $0 = b_3$ $b_3 = 0$

*time:* $\{t^0\} = \{t^{-a_3}\}$ $0 = -a_3$ $a_3 = 0$

*length:* $\{L^0\} = \{L^1L^{a_3}L^{-3b_3}L^{c_3}\}$ $0 = 1 + a_3 - 3b_3 + c_3$ $c_3 = -1$
$0 = 1 + c_3$

which yields

$\Pi_3$: $$\Pi_3 = \frac{y}{h}$$

**Step 6** We write the final functional relationship as

*Relationship between* Π*s*: $$\boxed{\frac{u}{V} = f\left(\text{Re}, \frac{y}{h}\right)} \qquad (2)$$

***Discussion*** We notice in the first and third Πs that when the parameter on which we are working has the same dimensions as one of the repeating parameters, the Π is simply the ratio of those two parameters (here *u*/*V* and *y*/*h*).

**7-57**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are seven parameters in this problem; $n = 7$,

*List of relevant parameters*: $u = f\left(\mu, V, h, \rho, y, t\right) \qquad n = 7 \qquad (1)$

**Step 2** The primary dimensions of each parameter are listed,

| $u$ | $\mu$ | $V$ | $h$ | $\rho$ | $y$ | $t$ |
|---|---|---|---|---|---|---|
| $\{L^1t^{-1}\}$ | $\{m^1L^{-1}t^{-1}\}$ | $\{L^1t^{-1}\}$ | $\{L^1\}$ | $\{m^1L^{-3}\}$ | $\{L^1\}$ | $\{t^1\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $j = 3$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected Πs*: $k = n - j = 7 - 3 = 4$

**Step 4** We need to choose three repeating parameters since $j = 3$. Following the guidelines outlined in this chapter, we do not pick the viscosity. It is better to pick a fixed length ($h$) rather than a variable length ($y$); otherwise $y$ would appear in each Pi, which would not be desirable. It would also not be wise to have time appear in each parameter. We choose

*Repeating parameters*: $V$, $\rho$, and $h$

**Step 5** The Πs are generated. The first three Πs are identical to those of the previous problem, so we do not include the details here. The fourth Π is formed by joining the new parameter $t$ to the repeating variables,

$$\Pi_4 = tV^{a_4}\rho^{b_4}h^{c_4} \qquad \{\Pi_4\} = \left\{\left(t^1\right)\left(L^1t^{-1}\right)^{a_4}\left(m^1L^{-3}\right)^{b_4}\left(L^1\right)^{c_4}\right\}$$

*mass*: $\{m^0\} = \{m^{b_4}\} \qquad 0 = b_4 \qquad b_4 = 0$

*time*: $\{t^0\} = \{t^1t^{-a_4}\} \qquad 0 = 1 - a_4 \qquad a_4 = 1$

*length*: $\{L^0\} = \{L^{a_1}L^{-3b_4}L^{c_4}\} \qquad 0 = a_4 - 3b_4 + c_4 \qquad c_4 = -1$

This Π is thus

$\Pi_4$: $$\Pi_4 = \frac{tV}{h}$$

**Step 6** Combining this result with the first three Πs from the previous problem,

*Relationship between Πs*: $$\boxed{\frac{u}{V} = f\left(\text{Re}, \frac{y}{h}, \frac{tV}{h}\right)} \qquad (2)$$

***Discussion*** As $t \to \infty$, $\Pi_4$ becomes irrelevant and the result degenerates into that of the previous problem.

**7-58**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are four parameters in this problem; $n$ = 4,

*List of relevant parameters*: $$c = f\left(k, T, R_{\text{gas}}\right) \qquad n = 4 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed; the ratio of specific heats $k$ is dimensionless.

| $c$ | $k$ | $T$ | $R_{\text{gas}}$ |
|---|---|---|---|
| $\{L^1 t^{-1}\}$ | $\{1\}$ | $\{T^1\}$ | $\{L^2 t^{-2} T^{-1}\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (T, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 4 - 3 = 1$$

Thus we expect only one Π.

**Step 4** We need to choose three repeating parameters since $j$ = 3. We only have one choice in this problem, since there are only three independent parameters on the right-hand side of Eq. 1. However, one of these is already dimensionless, so it is a Π all by itself. In this situation we reduce $j$ by one and continue,

*Reduction*: $$j = 3 - 1 = 2$$

If this revised value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 4 - 2 = 2$$

We now expect two Πs. We choose two repeating parameters since $j$ = 2,

*Repeating parameters*: $T$ and $R_{\text{gas}}$

**Step 5** The dependent Π is generated:

$$\Pi_1 = cT^{a_1} R_{\text{gas}}^{\ b_1} \qquad \{\Pi_1\} = \left\{\left(L^1 t^{-1}\right)\left(T^1\right)^{a_1}\left(L^2 t^{-2} T^{-1}\right)^{b_1}\right\}$$

*time*: $\{t^0\} = \{t^{-1-2b_1}\}$ $\qquad 0 = -1 - 2b_1 \qquad b_1 = -1/2$

*temperature*: $\{T^0\} = \{T^{a_1 - b_1}\}$ $\qquad a_1 = b_1 \qquad a_1 = -1/2$

*length*: $\{L^0\} = \{L^1 L^{2b_1}\}$ $\qquad 0 = 1 + 2b_1 \qquad b_1 = -1/2$

Fortunately the two results for exponent $b_1$ agree. The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{c}{\sqrt{R_{\text{gas}} T}}$$

The independent Π is already known,

$\Pi_2$: $$\Pi_2 = k$$

**Step 6** We write the final functional relationship as

*Relationship between* Π*s*: $$\boxed{\frac{c}{\sqrt{R_{\text{gas}}T}} = f(k)} \qquad (2)$$

***Discussion*** We cannot tell from dimensional analysis the exact form of the functional relationship. However, in this case the result agrees with the known equation for speed of sound in an ideal gas, $c = \sqrt{kR_{\text{gas}}T}$ .

**7-59**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are five parameters in this problem; $n = 5$,

*List of relevant parameters*: $c = f\left(k, T, R_u, M\right) \qquad n = 5$ (1)

**Step 2** The primary dimensions of each parameter are listed; the ratio of specific heats $k$ is dimensionless.

| $c$ | $k$ | $T$ | $R_u$ | $M$ |
|---|---|---|---|---|
| $\{L^1t^{-1}\}$ | $\{1\}$ | $\{T^1\}$ | $\{m^1L^2t^{-2}T^{-1}N^{-1}\}$ | $\{m^1N^{-1}\}$ |

**Step 3** As a first guess, $j$ is set equal to 5, the number of primary dimensions represented in the problem (m, T, L, N, and t).

*Reduction*: $j = 5$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $k = n - j = 5 - 5 = 0$

Obviously we cannot have zero Πs. We check that we have not missed a relevant parameter. Convinced that we have included all the relevant parameters we reduce $j$ by 1:

*Reduction*: $j = 5 - 1 = 4$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $k = n - j = 5 - 4 = 1$

**Step 4** We need to choose four repeating parameters since $j = 4$. We only have one choice in this problem, since there are only four independent parameters on the right-hand side of Eq. 1. However, one of these is already dimensionless, so it is a Π all by itself. In this situation we reduce $j$ by one (*again*) and continue,

*Reduction*: $j = 4 - 1 = 3$

If this revised value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $k = n - j = 5 - 3 = 2$

We now expect two Πs. Since $j = 3$ we choose three repeating parameters,

*Repeating parameters*: $T$, $M$, and $R_u$

**Step 5** The dependent Π is generated:

$$\Pi_1 = cT^{a_1}M^{b_1}R_u^{c_1} \qquad \{\Pi_1\} = \left\{\left(L^1t^{-1}\right)\left(T^1\right)^{a_1}\left(m^1n^{-1}\right)^{b_1}\left(m^1L^2t^{-2}T^{-1}N^{-1}\right)^{c_1}\right\}$$

*time*: $\{t^0\} = \{t^{-1}t^{-2c_1}\} \qquad 0 = -1 - 2c_1 \qquad c_1 = -1/2$

*mass*: $\{m^0\} = \{m^{b_1}m^{c_1}\} \qquad 0 = b_1 + c_1 \quad b_1 = -c_1 \qquad b_1 = 1/2$

| | | | |
|---|---|---|---|
| *amount of matter*: | $\left\{N^0\right\} = \left\{N^{-b_1} N^{-c_1}\right\}$ | $0 = -b_1 - c_1 \quad b_1 = -c_1$ | $b_1 = 1/2$ |
| *temperature*: | $\left\{T^0\right\} = \left\{T^{a_1} T^{-c_1}\right\}$ | $0 = a_1 - c_1 \quad a_1 = c_1$ | $a_1 = -1/2$ |
| *length*: | $\left\{L^0\right\} = \left\{L^1 L^{2c_1}\right\}$ | $0 = 1 + 2c_1$ | $c_1 = -1/2$ |

Fortunately the two results for exponent $b_1$ agree, and the two results for exponent $c_1$ agree. (If they did not agree, we would search for algebra mistakes. Finding none we would suspect that *j* is not correct or that we are missing a relevant parameter in the problem.) The dependent Π is thus

$\Pi_1$:
$$\Pi_1 = \frac{c\sqrt{M}}{\sqrt{R_u T}}$$

The independent Π is already known,

$\Pi_2$:
$$\Pi_2 = k$$

**Step 6** We write the final functional relationship as

*Relationship between* Π*s*:
$$\boxed{\Pi_1 = \frac{c\sqrt{M}}{\sqrt{R_u T}} = f(k)} \qquad (2)$$

***Discussion*** Since we know that $R_{gas} = R_u/M$, we see that the result here is the same as that of the previous problem.

**7-60**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are three parameters in this problem; $n = 3$,

*List of relevant parameters*: $c = f\left(T, R_{\text{gas}}\right) \qquad n = 3 \qquad (1)$

**Step 2** The primary dimensions of each parameter are listed,

| $c$ | $T$ | $R_{\text{gas}}$ |
|---|---|---|
| $\{L^1 t^{-1}\}$ | $\{T^1\}$ | $\{L^2 t^{-2} T^{-1}\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (T, L, and t).

*Reduction*: $j = 3$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected Πs*: $k = n - j = 3 - 3 = 0$

Obviously this is not correct, so we re-examine our initial assumptions. We can add another variable, $k$ (the ratio of specific heats) to our List of relevant parameters. This problem would then be identical to Problem 7-58. Instead, for instructional purposes we reduce $j$ by one and continue,

*Reduction*: $j = 3 - 1 = 2$

If this revised value of $j$ is correct, the expected number of Πs is

*Number of expected Πs*: $k = n - j = 3 - 2 = 1$

We now expect only one Π.

**Step 4** We need to choose two repeating parameters since $j = 2$. We only have one choice in this problem, since there are only two independent parameters on the right-hand side of Eq. 1,

*Repeating parameters*: $T$ and $R_{\text{gas}}$

**Step 5** The dependent Π is generated:

$$\Pi_1 = c T^{a_1} R_{\text{gas}}^{\ b_1} \qquad \{\Pi_1\} = \left\{ \left(L^1 t^{-1}\right) \left(T^1\right)^{a_1} \left(L^2 t^{-2} T^{-1}\right)^{b_1} \right\}$$

*time*: $\{t^0\} = \{t^{-1-2b_1}\} \qquad 0 = -1 - 2b_1 \qquad b_1 = -1/2$

*temperature*: $\{T^0\} = \{T^{a_1 - b_1}\} \qquad a_1 = b_1 \qquad a_1 = -1/2$

*length*: $\{L^0\} = \{L^1 L^{2b_1}\} \qquad 0 = 1 + 2b_1 \qquad b_1 = -1/2$

Fortunately the two results for exponent $b_1$ agree. The dependent Π is thus

$\Pi_1$:

$$\Pi_1 = \frac{c}{\sqrt{R_{\text{gas}}T}}$$

**Step 6** Since there is only one Π, it is a function of nothing. This is only possible if we set the Π equal to a constant. We write the final functional relationship as

*Relationship between* Π*s*:

$$\boxed{\Pi_1 = \frac{c}{\sqrt{R_{\text{gas}}T}} = \text{constant}} \qquad (2)$$

***Discussion*** Our result represents an interesting case of "luck". Although we failed to include the ratio of specific heats *k* in our analysis, we nevertheless obtain the correct result. In fact, if we set the constant in Eq. 2 as the square root of *k*, our result agrees with the known equation for speed of sound in an ideal gas, $c = \sqrt{kR_{\text{gas}}T}$ .

---

**7-61**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters, and compare to the known equation for an ideal gas.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are three parameters in this problem; $n$ = 3,

*List of relevant parameters*: $c = f\left(P, \rho\right) \qquad n = 3 \qquad (1)$

**Step 2** The primary dimensions of each parameter are listed,

$$\begin{matrix} c & P & \rho \\ \{L^1 t^{-1}\} & \{m^1 L^{-1} t^{-2}\} & \{m^1 L^{-3}\} \end{matrix}$$

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $j = 3$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected Πs*: $k = n - j = 3 - 3 = 0$

Obviously this is not correct, so we re-examine our initial assumptions. If we are convinced that $c$ is a function of only $P$ and $\rho$, we reduce $j$ by one and continue,

*Reduction*: $j = 3 - 1 = 2$

If this revised value of $j$ is correct, the expected number of Πs is

*Number of expected Πs*: $k = n - j = 3 - 2 = 1$

We now expect only one Π.

**Step 4** We need to choose two repeating parameters since $j$ = 2. We only have one choice in this problem, since there are only two independent parameters on the right-hand side of Eq. 1,

*Repeating parameters*: $P$ and $\rho$

**Step 5** The dependent Π is generated:

$$\Pi_1 = cP^{a_1}\rho^{b_1} \qquad \{\Pi_1\} = \left\{\left(L^1 t^{-1}\right)\left(m^1 L^{-1} t^{-2}\right)^{a_1}\left(m^1 L^{-3}\right)^{b_1}\right\}$$

*time*: $\{t^0\} = \{t^{-1} t^{-2a_1}\} \qquad 0 = -1 - 2a_1 \qquad a_1 = -1/2$

*mass*: $\{m^0\} = \{m^{a_1} m^{b_1}\} \qquad 0 = a_1 + b_1 \qquad b_1 = 1/2$

*length*: $\{L^0\} = \{L^1 L^{-2a_1} L^{-3b_1}\} \qquad \begin{matrix} 0 = 1 - a_1 - 3b_1 \\ 0 = 1 + \frac{1}{2} - \frac{3}{2} \end{matrix} \qquad 0 = 0$

Fortunately the exponents for length agree with those of mass and time. The dependent Π is thus

$\Pi_1$: $$\Pi_1 = c\sqrt{\frac{\rho}{P}}$$

**Step 6** Since there is only one Π, it is a function of nothing. This is only possible if we set the Π equal to a constant. We write the final functional relationship as

*Relationship between* Π*s*: $$\Pi_1 = c\sqrt{\frac{\rho}{P}} = \text{constant, or } c = \text{constant}\sqrt{\frac{P}{\rho}} \qquad (2)$$

The ideal gas equation is $P = \rho R_{gas}T$, or $P/\rho = R_{gas}T$. Thus, Eq. 2 can be written as

*Alternative result using ideal gas law*: $$c = \text{constant}\sqrt{R_{gas}T} \qquad (3)$$

Equation 3 is indeed consistent with the equation $c = \sqrt{kR_{gas}T}$ .

***Discussion*** There is no way to obtain the value of the constant in Eq. 2 or 3 solely by dimensional analysis, but it turns out that the constant is the square root of $k$.

**7-62**

**Solution** We are to use dimensional analysis to find a functional relationship between $F_D$ and variables $V$, $L$, and $\mu$.

***Assumptions*** **1** We assume Re << 1 so that the creeping flow approximation applies. **2** Gravitational effects are irrelevant. **3** No parameters other than those listed in the problem statement are relevant to the problem.

***Analysis*** We follow the step-by-step method of repeating variables.

**Step 1** There are four variables and constants in this problem; $n$ = 4. They are listed in functional form, with the dependent variable listed as a function of the independent variables and constants:

*List of relevant parameters*: $$F_D = f\left(V, L, \mu\right) \qquad n = 4$$

**Step 2** The primary dimensions of each parameter are listed.

| $F_D$ | $V$ | $L$ | $\mu$ |
|---|---|---|---|
| $\{m^1L^1t^{-2}\}$ | $\{L^1t^{-1}\}$ | $\{L^1\}$ | $\{m^1L^{-1}t^{-1}\}$ |

**Step 3** As a first guess, we set $j$ equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the number of Πs expected is

*Number of expected Πs*: $$k = n - j = 4 - 3 = 1$$

**Step 4** Now we need to choose three repeating parameters since $j$ = 3. Since we cannot choose the dependent variable, our only choices are $V$, $L$, and $\mu$.

**Step 5** Now we combine these repeating parameters into a product with the dependent variable $F_D$ to create the dependent Π,

*Dependent* Π: $$\Pi_1 = F_D V^{a_1} L^{b_1} \mu^{c_1} \qquad (1)$$

We apply the primary dimensions of Step 2 into Eq. 1 and force the Π to be dimensionless,

*Dimensions of* $\Pi_1$:
$$\{\Pi_1\} = \{m^0L^0t^0\} = \{F_D V^{a_1} L^{b_1} \mu^{c_1}\} = \left\{\left(m^1L^1t^{-2}\right)\left(L^1t^{-1}\right)^{a_1}\left(L^1\right)^{b_1}\left(m^1L^{-1}t^{-1}\right)^{c_1}\right\}$$

Now we equate the exponents of each primary dimension to solve for exponents $a_1$ through $c_1$.

*mass:* $\{m^0\} = \{m^1m^{c_1}\}$ $\quad 0 = 1 + c_1 \quad$ $c_1 = -1$

*time:* $\{t^0\} = \{t^{-2}t^{-a_1}t^{-c_1}\}$ $\quad 0 = -2 - a_1 - c_1 \quad$ $a_1 = -1$

*length:* $\{L^0\} = \{L^1L^{a_1}L^{b_1}L^{-c_1}\}$ $\quad 0 = 1 + a_1 + b_1 - c_1 \quad$ $b_1 = -1$

Equation 1 thus becomes

$\Pi_1$: $$\Pi_1 = \frac{F_D}{\mu V L} \qquad (2)$$

**Step 6** We now write the functional relationship between the nondimensional parameters. In the case at hand, there is only one Π, which is a function of *nothing*. This is possible only if the Π is constant. Putting Eq. 2 into standard functional form,

*Relationship between* Πs: $$\Pi_1 = \frac{F_D}{\mu V L} = f\left(\text{nothing}\right) = \text{constant} \qquad (3)$$

or

*Result of dimensional analysis*: $$\boxed{F_D = \text{constant} \cdot \mu V L} \qquad (4)$$

Thus we have shown that for creeping flow around an object, the aerodynamic drag force is simply a constant multiplied by $\mu VL$, regardless of the shape of the object.

***Discussion*** This result is very significant because all that is left to do is find the constant, which will be a function of the shape of the object (and its orientation with respect to the flow).

**7-63**

**Solution** We are to find the functional relationship between the given parameters and name any established dimensionless parameters.

***Assumptions*** **1** The given parameters are the only ones relevant to the flow at hand.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are five parameters in this problem; $n = 5$,

*List of relevant parameters*: $$V = f\left(d_p, \left(\rho_p - \rho\right), \mu, g\right) \qquad n = 5 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $V$ | $d_p$ | $(\rho_p - \rho)$ | $\mu$ | $g$ |
|---|---|---|---|---|
| $\{L^1t^{-1}\}$ | $\{L^1\}$ | $\{m^1L^{-3}\}$ | $\{m^1L^{-1}t^{-1}\}$ | $\{L^1t^{-2}\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected Πs*: $$k = n - j = 5 - 3 = 2$$

**Step 4** We need to choose three repeating parameters since $j = 3$. We pick length scale $d_p$, density difference ($\rho_p$ - $\rho$), and gravitational constant $g$.

*Repeating parameters*: $$d_p,\ \left(\rho_p \text{-} \rho\right), \text{ and } g$$

**Step 5** The Πs are generated. Note that for the first Π we do the algebra in our heads since the relationship is very simple. The dependent Π is

$\Pi_1$ = *a Froude number*: $$\Pi_1 = \frac{V}{\sqrt{gd_p}}$$

This Π is a type of **Froude number**. Similarly, the Π formed with viscosity is generated,

$$\Pi_2 = \mu d_p{}^a\left(\rho_p - \rho\right)^b g^c \qquad \{\Pi_2\} = \left\{\left(m^1L^{-1}t^{-1}\right)\left(L^1\right)^a\left(m^1L^{-3}\right)^b\left(L^1t^{-2}\right)^c\right\}$$

*mass:* $\{m^0\} = \{m^1m^b\}$ $\qquad 0 = 1 + b \qquad b = -1$

*time:* $\{t^0\} = \{t^{-1}t^{-2c}\}$ $\qquad 0 = -1 - 2c \qquad c = -\frac{1}{2}$

*length:* $\{L^0\} = \{L^{-1}L^aL^{-3b}L^c\}$ $\qquad 0 = -1 + a - 3b + c$, $\ 0 = -1 + a + 3 - \frac{1}{2} \qquad a = -\frac{3}{2}$

which yields

$\Pi_2$: $$\Pi_2 = \frac{\mu}{\left(\rho_p - \rho\right) d_p^{\frac{3}{2}} \sqrt{g}}$$

We recognize this Π as the inverse of a kind of **Reynolds number** if we split the $d_p$ terms to separate them into a length scale and (when combined with $g$) a velocity scale. The final form is

*Modified* $\Pi_2$ = *a Reynolds number*: $$\Pi_2 = \frac{\left(\rho_p - \rho\right) d_p \sqrt{g d_p}}{\mu}$$

**Step 6** We write the final functional relationship as

*Relationship between* Π*s*: $$\frac{V}{\sqrt{g d_p}} = f\left(\frac{\left(\rho_p - \rho\right) d_p \sqrt{g d_p}}{\mu}\right) \qquad (2)$$

***Discussion*** We cannot determine the *form* of the relationship by purely dimensional reasoning since there are two Πs. However, in Chap. 10 we shall see that $\Pi_1$ is a constant times $\Pi_2$.

**7-64**

**Solution** We are to develop an equation for the settling speed of an aerosol particle falling in air under creeping flow conditions.

***Assumptions*** **1** The particle falls at steady speed $V$. **2** The Reynolds number is small enough that the creeping flow approximation is valid.

***Analysis*** We start by recognizing that as the particle falls at steady settling speed, its net weight $W$ must equal the aerodynamic drag $F_D$ on the particle. We also know that $W$ is proportional to $(\rho_p - \rho)gd_p^3$. Thus,

*Equating forces*:
$$W = \text{constant}_1\left(\rho_p - \rho\right)gd_p^{\,3} = F_D = \text{constant}_2\,\mu V d_p \qquad (1)$$

where we have converted the notation of the previous problem, and we have defined two different constants. The two constants in Eq. 1 can be combined into one new constant for simplicity. Solving for $V$,

*Settling speed*:
$$\boxed{V = \text{constant}\frac{\left(\rho_p - \rho\right)gd_p^{\,2}}{\mu}} \qquad (2)$$

If we divide both sides of Eq. 2 by $\sqrt{gd_p}$ we see that the functional relationship given by Eq. 2 of the previous problem is consistent.

***Discussion*** This result is valid only if the Reynolds number is much smaller than one, as will be discussed in Chap. 10. If the particle is less dense than the fluid (e.g. bubbles rising in water), our result is still valid, but the particle rises instead of falls.

**7-65**

**Solution** We are to determine how the settling speed of an aerosol particle falling in air under creeping flow conditions changes when certain parameters are doubled.

***Assumptions*** **1** The particle falls at steady speed $V$. **2** The Reynolds number is small enough that the creeping flow approximation is valid.

***Analysis*** From the results of the previous problem, we see that **if particle size doubles, the settling speed increases by a factor of $2^2$ = 4**. Similarly, **if density difference doubles, the settling speed increases by a factor of $2^1$ = 2**.

***Discussion*** This result is valid only if the Reynolds number remains much smaller than unity, as will be discussed in Chap. 10. As the particle's settling speed increases by a factor of 2 or 4, the Reynolds number will also increase by that same factor. If the new Reynolds number is not small enough, the creeping flow approximation will be invalid and our results will not be correct, although the error will probably be small.

**7-66**

**Solution** We are to generate a nondimensional relationship between the given parameters.

***Assumptions*** **1** The flow is fully developed. **2** The fluid is incompressible. **3** No other parameters are significant in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters.

**Step 1** All the relevant parameters in the problem are listed in functional form:

*List of relevant parameters*: $\Delta P = f\left(V, \varepsilon, \rho, \mu, D, L\right) \qquad n = 7$

**Step 2** The primary dimensions of each parameter are listed:

| $\Delta P$ | $V$ | $\varepsilon$ | $\rho$ | $\mu$ | $D$ | $L$ |
|---|---|---|---|---|---|---|
| $\{m^1L^{-1}t^{-2}\}$ | $\{L^1t^{-1}\}$ | $\{L^1\}$ | $\{m^1L^{-3}\}$ | $\{m^1L^{-1}t^{-1}\}$ | $\{L^1\}$ | $\{L^1\}$ |

**Step 3** As a first guess, *j* is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $j = 3$

If this value of *j* is correct, the expected number of Πs is

*Number of expected* Π*s*: $k = n - j = 7 - 3 = 4$

**Step 4** We need to choose three repeating parameters since *j* = 3. Following the guidelines listed in Table 7-3, we cannot pick the dependent variable, Δ*P*. We cannot choose any two of parameters *ε*, *L*, and *D* since their dimensions are identical. It is not desirable to have *μ* or *ε* appear in all the Πs. The best choice of repeating parameters is thus *V*, *D*, and *ρ*.

*Repeating parameters*: *V*, *D*, and *ρ*

**Step 5** The dependent Π is generated:

$$\Pi_1 = \Delta P V^{a_1} D^{b_1} \rho^{c_1} \qquad \{\Pi_1\} = \left\{\left(m^1L^{-1}t^{-2}\right)\left(L^1t^{-1}\right)^{a_1}\left(L^1\right)^{b_1}\left(m^1L^{-3}\right)^{c_1}\right\}$$

*mass*: $\{m^0\} = \{m^1m^{c_1}\} \qquad 0 = 1 + c_1 \qquad c_1 = -1$

*time*: $\{t^0\} = \{t^{-2}t^{-a_1}\} \qquad 0 = -2 - a_1 \qquad a_1 = -2$

*length*: $\{L^0\} = \{L^{-1}L^{a_1}L^{b_1}L^{-3c_1}\} \qquad \begin{array}{l} 0 = -1 + a_1 + b_1 - 3c_1 \\ 0 = -1 - 2 + b_1 + 3 \end{array} \qquad b_1 = 0$

The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{\Delta P}{\rho V^2}$$

From Table 7-5, the established nondimensional parameter most similar to our $\Pi_1$ is the **Euler number** Eu. No manipulation is required.

We form the second Π with *μ*. By now we know that we will generate a **Reynolds number**,

$$\Pi_2 = \mu V^{a_2} D^{b_2} \rho^{c_2} \qquad \Pi_2 = \frac{\rho V D}{\mu} = \text{Reynolds number} = \text{Re}$$

The final two Π groups are formed with $\varepsilon$ and then with $L$. The algebra is trivial for these cases since their dimension (length) is identical to that of one of the repeating variables ($D$). The results are

$$\Pi_3 = \varepsilon V^{a_3} D^{b_3} \rho^{c_3} \qquad \Pi_3 = \frac{\varepsilon}{D} = \text{Roughness ratio}$$

$$\Pi_4 = L V^{a_4} D^{b_4} \rho^{c_4} \qquad \Pi_4 = \frac{L}{D} = \text{Length-to-diameter ratio or aspect ratio}$$

**Step 6** We write the final functional relationship as

*Relationship between* Π*s:*

$$\boxed{\text{Eu} = \frac{\Delta P}{\rho V^2} = f\left(\text{Re}, \frac{\varepsilon}{D}, \frac{L}{D}\right)} \qquad (1)$$

***Discussion*** The result applies to both laminar and turbulent fully developed pipe flow; it turns out, however, that the second independent Π (roughness ratio) is not nearly as important in laminar pipe flow as in turbulent pipe flow. Since Δ$P$ drops linearly with distance down the pipe, we know that Δ$P$ is linearly proportional to $L/D$. It is not possible to determine the functional relationships between the other Πs by dimensional reasoning alone.

**7-67**

**Solution** We are to determine by what factor volume flow rate increases in the case of fully developed laminar pipe flow when pipe diameter is doubled.

***Assumptions*** **1** The flow is steady. **2** The flow is fully developed, meaning that $dP/dx$ is constant and the velocity profile does not change downstream.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters.

**Step 1** All the relevant parameters in the problem are listed in functional form:

*List of relevant parameters*: $\dot{V} = f\left(D, \mu, \frac{dP}{dx}\right) \qquad n = 4$

**Step 2** The primary dimensions of each parameter are listed:

| $\dot{V}$ | $D$ | $\mu$ | $dP/dx$ |
|---|---|---|---|
| $\{L^3 t^{-1}\}$ | $\{L^1\}$ | $\{m^1 L^{-1} t^{-1}\}$ | $\{m^1 L^{-2} t^{-2}\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $j = 3$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected Πs*: $k = n - j = 4 - 3 = 1$

**Step 4** We need to choose three repeating parameters since $j = 3$. Here we must pick all three independent parameters,

*Repeating parameters*: $D$, $\mu$, and $dP/dx$

**Step 5** The Π is generated:

$$\Pi_1 = \dot{V}\left(dP/dx\right)^{a_1} D^{b_1} \mu^{c_1} \qquad \{\Pi_1\} = \left\{\left(L^3 t^{-1}\right)\left(m^1 L^{-2} t^{-2}\right)^{a_1}\left(L^1\right)^{b_1}\left(m^1 L^{-1} t^{-1}\right)^{c_1}\right\}$$

*mass*: $\{m^0\} = \{m^{a_1} m^{c_1}\} \qquad 0 = a_1 + c_1 \qquad c_1 = -a_1$

*time*: $\{t^0\} = \{t^{-1} t^{-2a_1} t^{-c_1}\}$

$$0 = -1 - 2a_1 - c_1 \qquad a_1 = -1$$
$$0 = -1 - 2a_1 + a_1 \qquad c_1 = 1$$

*length*: $\{L^0\} = \{L^3 L^{-2a_1} L^{b_1} L^{-c_1}\}$

$$0 = 3 - 2a_1 + b_1 - c_1 \qquad b_1 = -4$$
$$b_1 = -3 + 2a_1 + c_1$$

The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{\dot{V}\mu}{D^4 \frac{dP}{dx}}$$

**Step 6** Since there is only one Π, we set it equal to a constant. We write the final functional relationship as

*Relationship between Πs:*

$$\Pi_1 = \text{constant}, \qquad \dot{V} = \text{constant}\frac{D^4}{\mu}\frac{dP}{dx} \qquad (1)$$

We see immediately that **if the pipe diameter is doubled with all other parameters fixed, the volume flow rate will increase by a factor of** $2^4 = 16$.

***Discussion*** We will see in Chap. 9 that the constant is π/8. There is no way to obtain the value of the constant from dimensional analysis alone.

**7-68**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are four parameters in this problem; $n = 4$,

*List of relevant parameters*:
$$\dot{Q} = f\left(\dot{m}, c_p, \left(T_{\text{out}} - T_{\text{in}}\right)\right) \qquad n = 4 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $\dot{Q}$ | $\dot{m}$ | $c_p$ | $T_{\text{out}}$- $T_{\text{in}}$ |
|---|---|---|---|
| $\{m^1L^2t^{-3}\}$ | $\{m^1t^{-1}\}$ | $\{L^2t^{-2}T^{-1}\}$ | $\{T^1\}$ |

**Step 3** As a first guess, $j$ is set equal to 4, the number of primary dimensions represented in the problem (m, T, L, and t).

*Reduction*:
$$j = 4$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*:
$$k = n - j = 4 - 4 = 0$$

Obviously this is not correct, so we re-examine our initial assumptions. We are convinced that our list of parameters is sufficient, so we reduce $j$ by one and continue,

*Reduction*:
$$j = 4 - 1 = 3$$

If this revised value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*:
$$k = n - j = 4 - 3 = 1$$

We now expect only one Π.

**Step 4** We need to choose three repeating parameters since $j = 3$. We only have one choice in this problem, since there are only three independent parameters on the right-hand side of Eq. 1,

*Repeating parameters*:
$$\dot{m},\ c_p, \text{ and } \left(T_{\text{out}} - T_{\text{in}}\right)$$

**Step 5** The dependent Π is generated:

$$\Pi_1 = \dot{Q}\dot{m}^{a_1}c_p^{\,b_1}\left(T_{\text{out}} - T_{\text{in}}\right)^{c_1} \qquad \{\Pi_1\} = \left\{\left(m^1L^2t^{-3}\right)\left(m^1t^{-1}\right)^{a_1}\left(L^2t^{-2}T^{-1}\right)^{b_1}\left(T^1\right)^{c_1}\right\}$$

*mass*: $\{m^0\} = \{m^{1+a_1}\}$ $\qquad 0 = 1 + a_1 \qquad a_1 = -1$

*length*: $\{L^0\} = \{L^2L^{2b_1}\}$ $\qquad 0 = 2 + 2b_1 \qquad b_1 = -1$

*temperature*: $\{T^0\} = \{T^{-b_1+c_1}\}$ $\qquad c_1 = b_1 \qquad c_1 = -1$

*time*: $\{t^0\} = \{t^{-3-a_1-2b_1}\}$ $\qquad 0 = -3 - a_1 - 2b_1 \qquad 3 = 1 + 2$

Fortunately the result for the time exponents is consistent with that of the other dimensions. The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{\dot{Q}}{\dot{m}c_p\left(T_{\text{out}} - T_{\text{in}}\right)}$$

**Step 6** Since there is only one Π, it is a function of nothing. This is only possible if we set the Π equal to a constant. We write the final functional relationship as

*Relationship between Πs:* $$\boxed{\Pi_1 = \frac{\dot{Q}}{\dot{m}c_p\left(T_{\text{out}} - T_{\text{in}}\right)} = \text{constant}} \qquad (2)$$

***Discussion*** When there is only one Π, we know the functional relationship to within some (unknown) constant. In this particular case, comparing to Eq. 1 of Problem 7-24, we see that the constant is unity, $\dot{Q} = \dot{m}c_p\left(T_{\text{out}} - T_{\text{in}}\right)$. There is no way to obtain the constant in Eq. 2 from dimensional analysis; however, *one* experiment would be sufficient to determine the constant.

---

## Experimental Testing and Incomplete Similarity

### 7-69C

**Solution** We are to define wind tunnel blockage and discuss its acceptable limit. We are also to discuss the source of measurement errors at high values of blockage.

***Analysis*** Wind tunnel blockage is defined as the **ratio of model frontal area to cross-sectional area of the test-section**. The rule of thumb is that the blockage should be no more than 7.5%. If the blockage were significantly higher than this value, the flow would have to accelerate around the model much more than if the model were in an unbounded situation. Hence, similarity would not be achieved. We might expect the aerodynamic drag on the model to be too high since the *effective* freestream speed is too large due to the blockage.

***Discussion*** There are formulas to correct for wind tunnel blockage, but they become less and less reliable as blockage increases.

---

### 7-70C

**Solution** We are to discuss the rule of thumb concerning Mach number and incompressibility.

***Analysis*** **The rule of thumb is that the Mach number must stay below about 0.3 in order for the flow field to be considered "incompressible"**. What this really means is that compressibility effects, although present at all Mach numbers, are negligibly small compared to other effects driving the flow. If Ma is larger than about 0.3 in a wind tunnel test, the model flow field loses both kinematic and dynamic similarity, and the measured results are questionable. Of course, the error increases as Ma increases.

***Discussion*** Compressible flow is discussed in detail in Chap. 12. There you will see where the value 0.3 comes from.

---

## 7-71C

**Solution** We are to discuss some situations in which a model should be larger than its prototype.

***Analysis*** There are many possible situations, and students' examples should vary. Generally, any flow field that is very small and/or very fast benefits from simulation with a larger model. In most cases these are situations in which we want the model to be larger and slower so that experimental measurements and flow visualization are easier. Here are a few examples:

- **Modeling a hard disk drive.**
- **Modeling insect flight.**
- **Modeling the settling of very small particles in air or water.**
- **Modeling the motion of water droplets in clouds.**
- **Modeling flow through very fine tubing.**
- **Modeling biological systems like blood flow through capillaries, flow in the bronchi of lungs, etc.**

***Discussion*** You can think of several more examples.

---

## 7-72C

**Solution** We are to discuss the purpose of a moving ground belt and suggest an alternative.

***Analysis*** From the frame of reference of a moving car, both the air and the ground approach the car at freestream speed. When we test a model car in a wind tunnel, the air approaches at freestream speed, but the ground (floor of the wind tunnel) is *stationary*. Therefore we are not modeling the same flow. A boundary layer builds up on the wind tunnel floor, and the flow under the car cannot be expected to be the same as that under a real car. A moving ground belt solves this problem. Another way to say the same thing is to say that without the moving ground belt, there would not be kinematic similarity between the underside of the model and the underside of the prototype.

If a moving ground belt is unavailable, we could instead **install a false wall – i.e., a thin flat plate just above the boundary layer on the floor of the wind tunnel**. A sketch is shown in Fig. 1. At least then the boundary layer will be very thin and will not have as much influence on the flow under the model.

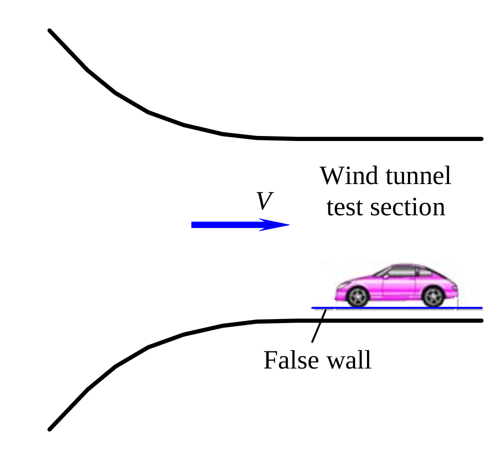



**FIGURE 1**
A false wall along the floor of a wind tunnel to reduce the size of the ground boundary layer.

***Discussion*** We discuss boundary layer growth in Chap. 10.

---

**7-73**

**Solution** We are to show that Froude number and Reynolds number are the dimensionless parameters that appear in a problem involving shallow water waves.

***Assumptions*** **1** Wave speed $c$ is a function only of depth $h$, gravitational acceleration $g$, fluid density $\rho$, and fluid viscosity $\mu$.

***Analysis*** We perform a dimensional analysis using the method of repeating variables.

**Step 1** There are five parameters in this problem; $n = 5$,

*List of relevant parameters*: $$c = f\left(h, \rho, \mu, g\right) \qquad n = 5$$

**Step 2** The primary dimensions of each parameter are listed,

| $c$ | $h$ | $\rho$ | $\mu$ | $g$ |
|---|---|---|---|---|
| $\{L^1t^{-1}\}$ | $\{L^1\}$ | $\{m^1L^{-3}\}$ | $\{m^1L^{-1}t^{-1}\}$ | $\{L^1t^{-2}\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 5 - 3 = 2$$

**Step 4** We need to choose three repeating parameters since $j$ = 3. We pick length scale $h$, density difference $\rho$, and gravitational constant $g$.

*Repeating parameters*: $h$, $\rho$, and $g$

**Step 5** The Πs are generated. Note that for the first Π we do the algebra in our heads since the relationship is very simple. The dependent Π is

$\Pi_1$ = *Froude number*: $$\Pi_1 = \text{Fr} = \frac{c}{\sqrt{gh}} \qquad (1)$$

This Π is the *Froude number*. Similarly, the Π formed with viscosity is generated,

$$\Pi_2 = \mu h^a \rho^b g^c \qquad \{\Pi_2\} = \left\{\left(m^1L^{-1}t^{-1}\right)\left(L^1\right)^a\left(m^1L^{-3}\right)^b\left(L^1t^{-2}\right)^c\right\}$$

*mass:* $\{m^0\} = \{m^1m^b\}$ $\qquad 0 = 1 + b \qquad b = -1$

*time:* $\{t^0\} = \{t^{-1}t^{-2c}\}$ $\qquad 0 = -1 - 2c \qquad c = -\frac{1}{2}$

*length:* $\{L^0\} = \{L^{-1}L^aL^{-3b}L^c\}$ $\qquad 0 = -1 + a - 3b + c$, $0 = -1 + a + 3 - \tfrac{1}{2} \qquad a = -\frac{3}{2}$

which yields

$\Pi_2$: $$\Pi_2 = \frac{\mu}{\rho h^{\frac{3}{2}}\sqrt{g}}$$

We can manipulate this Π into the Reynolds number if we invert it and then multiply by Fr (Eq. 1) The final form is

*Modified* $\Pi_2$ = *Reynolds number*: $$\Pi_2 = \text{Re} = \frac{\rho ch}{\mu}$$

**Step 6** We write the final functional relationship as

*Relationship between Πs*: $$\text{Fr} = \frac{c}{\sqrt{gh}} = f\left(\text{Re}\right) \quad \text{where} \quad \text{Re} = \frac{\rho ch}{\mu}$$

***Discussion*** As discussed in this chapter, it is often difficult to match both Fr and Re between a model and a prototype.

---

## 7-74

**Solution** We are to nondimensionalize experimental pipe data, plot the data, and determine if Reynolds number independence has been achieved. We are then to extrapolate to a higher speed.

***Assumptions*** **1** The flow is fully developed. **2** The flow is steady and incompressible.

***Properties*** For water at $T$ = 20°C and atmospheric pressure, $\rho$ = 998.0 kg/m³ and $\mu$ = 1.002 × 10⁻³ kg/m·s.

***Analysis*** (*a*) We convert each data point in the table from $V$ and $\Delta P$ to Reynolds number and Euler number. The calculations at the last (highest speed) data point are shown here:

*Reynolds number*:

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{\left(998.0\text{ kg/m}^3\right)\left(50\text{ m/s}\right)\left(0.104\text{ m}\right)}{1.002\times10^{-3}\text{ kg/m}\cdot\text{s}} = 5.18\times10^6 \quad (1)$$

and

*Euler number*:

$$\text{Eu} = \frac{\Delta P}{\rho V^2} = \frac{758{,}700\text{ N/m}^2}{\left(998.0\text{ kg/m}^3\right)\left(50\text{ m/s}\right)^2}\left(\frac{\text{kg m}}{\text{s}^2\text{N}}\right) = 0.304 \quad (2)$$

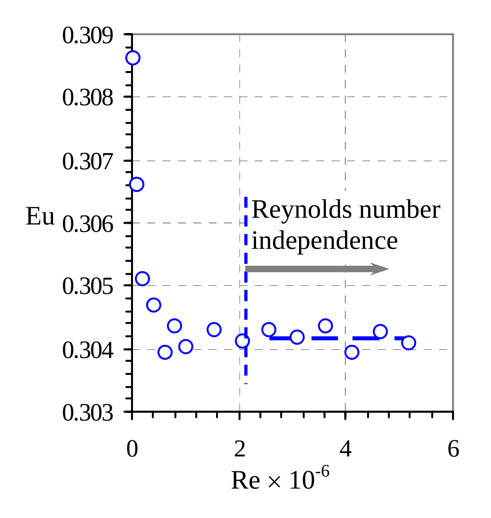



**FIGURE 1**
Nondimensionalized experimental data from a section of pipe.

We plot Eu versus Re in Fig. 1. Although there is experimental scatter in the data, it appears that Reynolds number independence has been achieved beyond a Reynolds number of about 2 × 10⁶. The average value of Eu based on the last 6 data points is 0.3042.

(*b*) We extrapolate to higher speeds. At $V$ = 80 m/s, we calculate $\Delta P$, assuming that Eu remains constant to higher values of Re,

*Extrapolated value*:

$$\Delta P = \text{Eu}\times\rho V^2 = 0.3042\left(998.0\text{ kg/m}^3\right)\left(80\text{ m/s}\right)^2\left(\frac{\text{s}^2\text{N}}{\text{kg m}}\right) = \mathbf{1{,}940{,}000\ N/m^2} \quad (3)$$

***Discussion*** It is shown in Chap. 8 that Reynolds number independence is indeed achieved at high-enough values of Re. The threshold value above which Re independence is achieved is a function of relative roughness height, $\varepsilon/D$.

---

## 7-75

**Solution** We are to calculate the wind tunnel blockage of a model truck in a wind tunnel and determine if it is within acceptable limits.

***Assumptions*** **1** The frontal area is equal to truck width times height. (Note that the actual area of the truck may be somewhat smaller than this due to rounded corners and the air gap under the truck, but a truck looks nearly like a rectangle from the front, so this is not a bad approximation.)

***Analysis*** Wind tunnel blockage is defined as the ratio of model frontal area to cross-sectional area of the test-section,

*Blockage*:
$$Blockage = \frac{A_{\text{model}}}{A_{\text{wind tunnel}}} = \frac{(0.159\text{ m})(0.257\text{ m})}{(1.2\text{ m})(1.0\text{ m})} = 0.034 = \mathbf{3.4\%} \qquad (1)$$

The rule of thumb is that the blockage should be no more than 7.5%. Since we are well below this value, we need not worry about blockage effects.

***Discussion*** The *length* of the model does not enter our analysis since we are only concerned with the frontal area of the model.

---

## 7-76C

**Solution** We are to discuss whether Reynolds number independence has been achieved, and whether the researchers can be confident about it.

***Analysis*** We remove the last four data points from Table 7-7 and from Fig. 7-41. From the remaining data it appears that the drag coefficient is beginning to level off, but is still decreasing with Re. Thus, **the researchers do not know if they have achieved Reynolds number independence or not**.

***Discussion*** The wind tunnel speed is too low to achieve Reynolds number independence.

---

**7-77E**

**Solution** We are to calculate the size and scale of the model truck to be constructed, and calculate its maximum Reynolds number. Then we are to determine whether this model in this wind tunnel will achieve Reynolds number independence.

***Assumptions*** **1** The model will be constructed carefully so as to achieve approximate geometric similarity. **2** The wind tunnel air is at the same temperature and pressure as that flowing over the prototype truck.

***Properties*** For air at $T = 80^oF$ and atmospheric pressure, $\rho = 0.07350$ lbm/ft$^3$ and $\mu = 1.248 \times 10^{-5}$ lbm/ft·s.

***Analysis*** (*a*) The rule of thumb about blockage is that we should keep the blockage below 7.5%. Thus, the frontal area of the model truck must be no more than $0.075 \times A_{\text{wind tunnel}}$. The ratio of height to width of the full-scale truck is $H_p/W_p = 12/8.33 = 1.44$. Thus, for the geometrically similar model truck,

*Equation for model truck width*:
$$W_m = \frac{A_m}{H_m} = \frac{7.5\% A_{\text{wind tunnel}}}{1.44 W_m} \quad (1)$$

We solve Eq. 1 for $W_m$,

*Model truck width*:
$$W_m = \sqrt{\frac{7.5\% A_{\text{wind tunnel}}}{1.44}} = \sqrt{\frac{0.075\left(400 \text{ in}^2\right)}{1.44}} = 4.56 \text{ in} \quad (2)$$

Scaling the height and length geometrically,

*Model truck dimensions*:
$$W_m = \mathbf{4.56\ in},\ H_m = \mathbf{6.57\ in},\ L_m = \mathbf{28.5\ in} \quad (3)$$

These dimensions represent a model that is scaled at approximately 1:22.

(*b*) At the maximum speed, with Re based on truck width,

*Maximum* Re:
$$\text{Re} = \frac{\rho W_m V_{max}}{\mu} = \frac{\left(0.07350 \text{ lbm/ft}^3\right)\left(4.56 \text{ in}\right)\left(160 \text{ ft/s}\right)}{1.248\times 10^{-5} \text{ lbm/ft}\cdot\text{s}}\left(\frac{1 \text{ ft}}{12 \text{ in}}\right) = \mathbf{3.58\times 10^5} \quad (4)$$

(*c*) Based on the data of Fig. 7-41, **this Reynolds number is shy of the value needed to achieve Reynolds number independence**.

***Discussion*** The students should run at the highest wind tunnel speed. Their measured values of $C_D$ will probably be higher than those of the prototype, but the *relative* difference in $C_D$ due to their modifications should still be valid.

**7-78**

**Solution** We are to calculate and plot $C_D$ as a function of Re for a given set of wind tunnel measurements, and determine if dynamic similarity and/or Reynolds number independence have been achieved. Finally, we are to estimate the aerodynamic drag force acting on the prototype car.

***Assumptions*** **1** The model car is geometrically similar to the prototype car. **2** The aerodynamic drag on the strut holding the model car is negligible.

***Properties*** For air at atmospheric pressure and at $T$ = 25°C, $\rho$ = 1.184 kg/m³ and $\mu$ = 1.849 × 10⁻⁵ kg/(m s).

***Analysis*** We calculate $C_D$ and Re for the last data point listed in the given table (at the fastest wind tunnel speed),

*Model drag coefficient at last data point*:

$$C_{D,\text{m}} = \frac{F_{D,\text{m}}}{\frac{1}{2}\rho_\text{m} V_\text{m}^{\,2} A_\text{m}} = \frac{4.91\text{ N}}{\frac{1}{2}\left(1.184\text{ kg/m}^3\right)\left(55\text{ m/s}\right)^2 \frac{\left(1.69\text{ m}\right)\left(1.30\text{ m}\right)}{16^2}}\left(\frac{\text{kg m}}{\text{s}^2\text{N}}\right) = 0.319$$

and

*Model Reynolds number at last data point*:

$$\text{Re}_\text{m} = \frac{\rho_\text{m} V_\text{m} W_\text{m}}{\mu_\text{m}} = \frac{\left(1.184\text{ kg/m}^3\right)\left(55\text{ m/s}\right)\left(\frac{1.69}{16}\text{ m}\right)}{1.849\times10^{-5}\text{ kg/m}\cdot\text{s}} = 3.72\times10^5 \qquad (1)$$

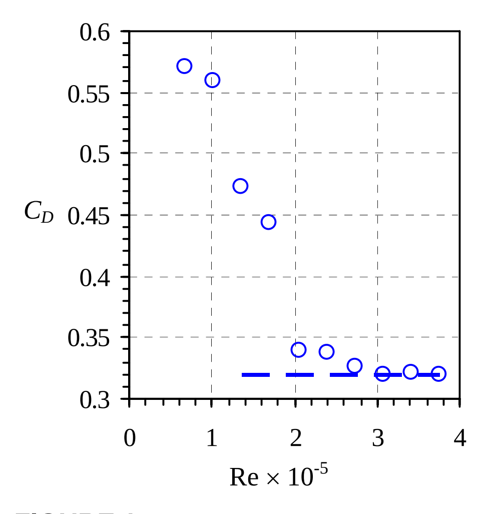



**FIGURE 1**
Aerodynamic drag coefficient as a function of Reynolds number – results nondimensionalized from wind tunnel test data on a model car.

We repeat the above calculations for all the data points in the given table, and we plot $C_D$ verses Re in Fig. 1.

Have we achieved dynamic similarity? Well, we have *geometric* similarity between model and prototype, but the Reynolds number of the prototype car is

*Reynolds number of prototype car*:
$$\text{Re}_\text{p} = \frac{\rho_\text{p} V_\text{p} W_\text{p}}{\mu_\text{p}} = \frac{\left(1.184\text{ kg/m}^3\right)\left(29\text{ m/s}\right)\left(1.69\text{ m}\right)}{1.849\times10^{-5}\text{ kg/m}\cdot\text{s}} = 3.14\times10^6 \qquad (2)$$

where the width and speed of the prototype are used in the calculation of $\text{Re}_\text{p}$. Comparison of Eqs. 1 and 2 reveals that the prototype Reynolds number is more than eight times larger than that of the model. Since we cannot match the independent Πs in the problem, **dynamic similarity has not been achieved**.

Have we achieved Reynolds number independence? From Fig. 1 we see that **Reynolds number independence has indeed been achieved** – at Re greater than about 3 × 10⁵, $C_D$ has leveled off to a value of about 0.32 (to two significant digits).

Since we have achieved Reynolds number independence, we can extrapolate to the full scale prototype, assuming that $C_D$ remains constant as Re is increased to that of the full scale prototype.

*Aerodynamic drag on the prototype*:

$$F_{D,\text{p}} = \frac{1}{2}\rho_\text{p} V_\text{p}^{\,2} A_\text{p} C_{D,\text{p}} = \frac{1}{2}\left(1.184\text{ kg/m}^3\right)\left(29\text{ m/s}\right)^2\left(1.69\text{ m}\right)\left(1.30\text{ m}\right)0.32\left(\frac{\text{s}^2\text{N}}{\text{kg m}}\right) = \mathbf{350\text{ N}}$$

***Discussion*** We give our final result to two significant digits.

## Review Problems

### 7-79C

**Solution**

(*a*) **False**: Kinematic similarity is a necessary but not sufficient condition for dynamic similarity.
(*b*) **True**: You cannot have dynamic similarity if the model and prototype are not geometrically similar.
(*c*) **True**: You cannot have kinematic similarity if the model and prototype are not geometrically similar.
(d) **False**: It is possible to have kinematic similarity (scaled velocities at corresponding points), yet not have dynamic similarity (forces do not scale at corresponding points).

### 7-80C

**Solution** We are to think of and describe a prototype and model flow in which there is geometric but not kinematic similarity even though $\text{Re}_m = \text{Re}_p$.

***Analysis*** Students' responses will vary. Here are some examples:

- A model car is being tested in a wind tunnel such that there is geometric similarity and the wind tunnel speed is adjusted so that $\text{Re}_m = \text{Re}_p$. However, there is not a moving ground belt, so there is not kinematic similarity between the model and prototype.
- A model airplane is being tested in a wind tunnel such that there is geometric similarity and the wind tunnel speed is adjusted so that $\text{Re}_m = \text{Re}_p$. However, the Mach numbers are quite different, and therefore kinematic similarity is not achieved.
- A model of a river or waterfall or other open surface flow problem in which there is geometric similarity and the speed is adjusted so that $\text{Re}_m = \text{Re}_p$. However, the Froude numbers do not match and therefore the velocity fields are not similar and kinematic similarity is not achieved.

***Discussion*** There are many more acceptable cases that students may imagine.

### 7-81C

**Solution** We are to find at least three established nondimensional parameters not listed in Table 7-5, and list these following the format of that table.

***Analysis*** Students' responses will vary. Here are some examples:

| Name | Definition | Ratio of significance |
|---|---|---|
| Bingham number | $\text{Bm} = \dfrac{\tau L}{\mu V}$ | $\dfrac{\text{yield stress}}{\text{viscous stress}}$ |
| Elasticity number | $\text{El} = \dfrac{t_c \mu}{\rho L^2}$ | $\dfrac{\text{elastic force}}{\text{inertial force}}$ |
| Galileo number | $\text{Ga} = \dfrac{g D^3 \rho^2}{\mu^2}$ | $\dfrac{\text{gravitational force}}{\text{viscous force}}$ |

In the above, $t_c$ is a characteristic time.

***Discussion*** There are many more established dimensionless parameters in the literature. Some sneaky students may make up their own!

**7-82**

**Solution** We are to determine the primary dimensions of each variable, and then show that Hooke's law is dimensionally homogeneous.

***Analysis***

(*a*) Moment of inertia has dimensions of length$^4$,

*Primary dimensions of moment of inertia*:
$$\{I\} = \{\text{length}^4\} = \{\mathbf{L^4}\} \qquad (1)$$

(*b*) Modulus of elasticity has the same dimensions as pressure,

*Primary dimensions of modulus of elasticity*:
$$\{E\} = \left\{\frac{\text{force}}{\text{area}}\right\} = \left\{\frac{\text{mass}\times\text{length}}{\text{time}^2}\times\frac{1}{\text{length}^2}\right\} = \left\{\frac{\mathbf{m}}{\mathbf{Lt^2}}\right\} \qquad (2)$$

Or, in exponent form, $\{E\}$ = $\{\mathbf{m^1\,L^{-1}\,t^{-2}}\}$.

(*c*) Strain is defined as change in length per unit length, so it is dimensionless.

*Primary dimensions of strain*:
$$\{\varepsilon\} = \left\{\frac{\text{length}}{\text{length}}\right\} = \{\mathbf{1}\} \qquad (3)$$

(*d*) Stress is force per unit area, again just like pressure.

*Primary dimensions of stress*:
$$\{\sigma\} = \left\{\frac{\text{force}}{\text{area}}\right\} = \left\{\frac{\text{mass}\times\text{length}}{\text{time}^2\times\text{length}^2}\right\} = \left\{\frac{\mathbf{m}}{\mathbf{Lt^2}}\right\} \qquad (4)$$

Or, in exponent form, $\{\sigma\}$ = $\{\mathbf{m^1\,L^{-1}\,t^{-2}}\}$.

(*e*) Hooke's law is $\sigma = E\varepsilon$. We write the primary dimensions of both sides:

*Primary dimensions of Hooke's law*:
$$\{\sigma\} = \left\{\frac{\mathbf{m}}{\mathbf{Lt^2}}\right\} = \{E\varepsilon\} = \left\{\frac{\text{m}}{\text{Lt}^2}\times 1\right\} = \left\{\frac{\mathbf{m}}{\mathbf{Lt^2}}\right\} \qquad (5)$$

Or, in exponent form, the dimensions of both sides of the equation are $\{\mathbf{m^1\,L^{-1}\,t^{-2}}\}$. Thus we see that **Hooke's law is indeed dimensionally homogeneous**.

***Discussion*** If the dimensions of Eq. 5 were not homogeneous, we would surely expect that we made an error somewhere.

**7-83**

**Solution** We are to find the functional relationship between the given parameters and name any established dimensionless parameters.

***Assumptions*** **1** The given parameters are the only ones relevant to the flow at hand.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are five parameters in this problem; $n = 5$,

*List of relevant parameters*: $$z_d = f\left(F, L, E, I\right) \qquad n = 5 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $z_d$ | $F$ | $L$ | $E$ | $I$ |
|---|---|---|---|---|
| $\{L^1\}$ | $\{m^1L^1t^{-2}\}$ | $\{L^1\}$ | $\{m^1L^{-1}t^{-2}\}$ | $\{L^4\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 5 - 3 = 2$$

**Step 4** We need to choose three repeating parameters since $j = 3$. We cannot pick both length $L$ and moment of inertia $I$ since their dimensions differ only by a power. We also notice that we cannot choose $F$, $L$, and $E$ since these three parameters can form a Π all by themselves. So, we set $j = 3 - 1 = 2$, and we choose two repeating parameters, expecting $5 - 2 = 3$ Πs,

*Repeating parameters*: $L$ and $E$

**Step 5** The Πs are generated. Note that for the first Π we do the algebra in our heads since $z_d$ has the same dimensions as $L$. The dependent Π is

$\Pi_1$: $$\Pi_1 = \frac{z_d}{L}$$

This Π is not an established dimensionless group, although it is a ratio of two lengths, **similar to an aspect ratio**.

We form the second Π with force $F$:

$$\Pi_2 = FL^aE^b \qquad \{\Pi_2\} = \left\{\left(m^1L^1t^{-2}\right)\left(L^1\right)^a\left(m^1L^{-1}t^{-2}\right)^b\right\}$$

*mass:* $\{m^0\} = \{m^1m^b\}$ $\qquad 0 = 1 + b \qquad b = -1$

*time:* $\{t^0\} = \{t^{-2}t^{-2b}\}$ $\qquad 0 = -2 - 2b \qquad b = -1$

*length:* $\{L^0\} = \{L^1L^aL^{-b}\}$ $\qquad 0 = 1 + a - b$, $a = -1 + b \qquad a = -2$

which yields

$\Pi_2$: $$\Pi_2 = \frac{F}{L^2 E}$$

We do not recognize $\Pi_2$ as a named dimensionless parameter.

The final Π is formed with moment of inertia. Since {I} = {$L^4$}, there is no need to go through the algebra – we write

$\Pi_3$: $$\Pi_3 = \frac{I}{L^4}$$

Again, we do not recognize $\Pi_2$ as a named dimensionless parameter.

**Step 6** We write the final functional relationship as

*Relationship between* Π*s*: $$\boxed{\frac{z_d}{L} = f\left(\frac{F}{L^2 E}, \frac{I}{L^4}\right)} \qquad (2)$$

***Discussion*** We cannot determine the *form* of the relationship by purely dimensional reasoning since there are three Πs.

**7-84**

**Solution** We are to generate dimensionless relationships among given parameters, and then we are to discuss how $\Delta P$ decreases if the time is doubled.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** (*a*) We perform dimensional analyses using the method of repeating variables. First we analyze $\Delta P$:

**Step 1** There are four parameters in this problem; $n = 4$,

*List of relevant parameters*: $$\Delta P = f(t, c, E) \qquad n = 4 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $\Delta P$ | t | $c$ | $E$ |
|---|---|---|---|
| $\{m^1 L^{-1} t^{-2}\}$ | $\{t^1\}$ | $\{L^1 t^{-1}\}$ | $\{m^1 L^2 t^{-2}\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 4 - 3 = 1$$

**Step 4** We need to choose three repeating parameters since $j = 3$. We pick all the independent parameters – time $t$, speed of sound $c$, and energy $E$,

*Repeating parameters*: $t$, $c$, and $E$

**Step 5** The dependent Π is generated:

$$\Pi_1 = \Delta P \times t^{a_1} c^{b_1} E^{c_1} \qquad \{\Pi_1\} = \left\{ \left(m^1 L^{-1} t^{-2}\right) \left(t^1\right)^{a_1} \left(L^1 t^{-1}\right)^{b_1} \left(m^1 L^2 t^{-2}\right)^{c_1} \right\}$$

*mass*: $$\{m^0\} = \{m^1 m^{c_1}\} \qquad 0 = 1 + c_1 \qquad c_1 = -1$$

*length*: $$\{L^0\} = \{L^{-1} L^{b_1} L^{2c_1}\} \qquad 0 = -1 + b_1 + 2c_1 \qquad b_1 = 3$$
$$b_1 = 1 - 2c_1$$

*time*: $$\{t^0\} = \{t^{-2} t^{a_1} t^{-b_1} t^{-2c_1}\} \qquad 0 = -2 + a_1 - b_1 - 2c_1 \qquad a_1 = 3$$
$$a_1 = 2 + b_1 + 2c_1$$

The dependent Π is thus

$\Pi_1$ for $\Delta P$: $$\Pi_1 = \frac{t^3 c^3 \Delta P}{E}$$

This Π is not an established one, so we leave it as is.

**Step 6** We write the final functional relationship as

*Relationship between* Π*s*: $$\boxed{\Delta P = \text{constant}\frac{E}{t^3 c^3}} \qquad (2)$$

We perform a similar dimensional analysis using the same repeating variables, but this time for radius *r*. We do not show the algebra since the Π can be found by inspection. We get

$\Pi_1$ for *r*: $$\Pi_1 = \frac{r}{ct}$$

Since this is the only Π, it must be equal to a constant,

*Relationship between* Π*s*: $$r = \text{constant} \cdot ct \qquad (3)$$

(*b*) From Eq. 2 we see that **if *t* is doubled, Δ*P* decreases by a factor of $2^3$ = 8**.

***Discussion*** The pressure rise across the blast wave decays rapidly with time (and with distance from the explosion). The speed of sound depends on temperature. If the explosion is of sufficient strength, *T* will increase significantly and *c* will not remain constant.

---

## 7-85

**Solution** We are to find an alternate definition of Archimedes number, and list it following the format of Table 7-5. Then we are to find an established Π group that is similar.

***Analysis*** Students' responses will vary. There seems to be a plethora of definitions of Archimedes number. Here is the one most appropriate for buoyant fluids:

| Name | Definition | Ratio of significance |
|---|---|---|
| Archimedes number | $\text{Ar} = \dfrac{gL\Delta\rho}{\rho V^2}$ | $\dfrac{\text{buoyant force}}{\text{inertial force}}$ |

In the above, $\Delta\rho$ is a characteristic density difference in the fluid (due to buoyancy) and $\rho$ is a characteristic or average density of the fluid. A glance through Table 7-5 shows that the **Richardson number** is very similar to this alternative definition of Ar. In fact, the alternate form of Ri (Problem 7-55) is *identical* to our new Ar.

***Discussion*** Some students may find other definitions that are also valid. For example, $\Delta\rho/\rho$ may be replaced by $\Delta T/T$.

---

**7-86**

**Solution** We are to generate a dimensionless relationship between the given parameters.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane. **3** The flow is fully developed.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are five parameters in this problem; $n = 5$,

*List of relevant parameters*: $$u = f\left(h, \frac{dP}{dx}, \mu, y\right) \qquad n = 5 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $u$ | $h$ | $dP/dx$ | $\mu$ | $y$ |
|---|---|---|---|---|
| $\{L^1t^{-1}\}$ | $\{L^1\}$ | $\{m^1L^{-2}t^{-2}\}$ | $\{m^1L^{-1}t^{-1}\}$ | $\{L^1\}$ |

**Step 3** As a first guess, *j* is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of *j* is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 5 - 3 = 2$$

**Step 4** We need to choose three repeating parameters since $j = 3$. We cannot pick both *h* and *y* since they have the same dimensions. We choose

*Repeating parameters*: *h*, *dP/dx*, and $\mu$

**Step 5** The dependent Π is generated:

$$\Pi_1 = uh^{a_1}\left(\frac{dP}{dx}\right)^{b_1}\mu^{c_1} \qquad \{\Pi_1\} = \left\{\left(L^1t^{-1}\right)\left(L^1\right)^{a_1}\left(m^1L^{-2}t^{-2}\right)^{b_1}\left(m^1L^{-1}t^{-1}\right)^{c_1}\right\}$$

*mass*: $$\{m^0\} = \{m^{b_1}m^{c_1}\} \qquad 0 = b_1 + c_1 \qquad c_1 = -b_1$$

*time*: $$\{t^0\} = \{t^{-1}t^{-2b_1}t^{-c_1}\} \qquad \begin{aligned} 0 &= -1 - 2b_1 - c_1 \\ 0 &= -1 - b_1 \end{aligned} \qquad \begin{aligned} b_1 &= -1 \\ c_1 &= 1 \end{aligned}$$

*length*: $$\{L^0\} = \{L^1L^{a_1}L^{-2b_1}L^{-c_1}\} \qquad \begin{aligned} 0 &= 1 + a_1 - 2b_1 - c_1 \\ 0 &= 1 + a_1 + 1 \end{aligned} \qquad a_1 = -2$$

The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{\mu u}{h^2 \frac{dP}{dx}}$$

The independent Π is generated with variable *y*. Since $\{y\} = \{L\}$, and this is the same as one of the repeating variables (*h*), $\Pi_2$ is simply $y/h$,

$\Pi_2$: $$\Pi_2 = \frac{y}{h}$$

**Step 6** We write the final functional relationship as

*Relationship between Πs:*

$$\boxed{\Pi_1 = \frac{\mu u}{h^2 \frac{dP}{dx}} = f\left(\frac{y}{h}\right)} \qquad (2)$$

***Discussion*** If we were to solve this problem exactly (using the methods of Chap. 9) we would see that the functional relationship of Eq. 2 is correct.

---

## 7-87

**Solution** We are to generate a dimensionless relationship between the given parameters and then analyze the behavior of $u_{max}$ when an independent variable is doubled.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane. **3** The flow is fully developed.

***Analysis*** (*a*) A step-by-step dimensional analysis procedure could be performed. However, we notice that $u_{max}$ has the same dimensions as *u*. Therefore the algebra would be identical to that of the previous problem except that there is only one Π instead of two since *y* is no longer a parameter. The result is

*Relationship between Πs:*

$$\Pi_1 = \frac{\mu u_{max}}{h^2 \frac{dP}{dx}} = \text{constant} = C \qquad (1)$$

or

*Final relationship for* $u_{max}$:

$$\boxed{u_{max} = C\frac{h^2}{\mu}\frac{dP}{dx}} \qquad (2)$$

Alternatively, we can use the results of the previous problem directly. Namely, since we know that the maximum velocity occurs at the centerline, $y/h$ = 1/2 there, and is a constant. Hence, Eq. 2 of the previous problem reduces to Eq. 1 of the present problem.

(*b*) If *h* doubles, we see from Eq. 2 that $u_{max}$ will increase by a factor of $2^2 = 4$.

(*c*) If $dP/dx$ doubles, we see from Eq. 2 that $u_{max}$ will increase by a factor of $2^1 = 2$.

(*d*) Since there is only one Π in this problem, we would need to conduct only *one experiment* to determine the constant *C* in Eq. 2.

***Discussion*** The constant turns out to be -1/8, but there is no way to determine this from dimensional analysis alone. To obtain the constant, we would need to either do an experiment, or solve the problem exactly using the methods discussed in Chap. 9.

---

**7-88** [*Also solved using EES on enclosed DVD*]

**Solution** We are to generate a relationship for Darcy friction factor $f$ in terms of Euler number Eu. We are then to plot $f$ as a function of Re and discuss whether Reynolds number independence has been achieved.

***Assumptions*** **1** The flow is fully developed. **2** The flow is steady and incompressible.

***Properties*** For water at $T$ = 20°C and atmospheric pressure, $\rho$ = 998.0 kg/m$^3$ and $\mu$ = 1.002 × 10$^{-3}$ kg/m·s.

***Analysis*** (*a*) Since the flow is fully developed, the control volume cuts through two cross sections in which the velocity profiles are identical. The flow is also steady, so the control volume momentum equation in the horizontal ($x$) direction reduces to

*Conservation of momentum*:

$$\sum F_x = \sum F_{x,\text{ pressure}} + \sum F_{x,\text{ shear stress}} = 0 \qquad (1)$$

We multiply pressure by cross-sectional area to obtain the pressure force, and wall shear stress times inner pipe wall surface area to obtain the shear stress force,

$$\sum F_{x,\text{ pressure}} = \Delta P \frac{\pi D^2}{4} \qquad \sum F_{x,\text{ shear stress}} = -\tau_w \pi D L \qquad (2)$$

Note the negative sign in the shear stress term since $\tau_w$ points to the left. We substitute Eq. 2 into Eq. 1. After some algebra,

*Result*:
$$\Delta P = \frac{4\tau_w L}{D} \qquad (3)$$

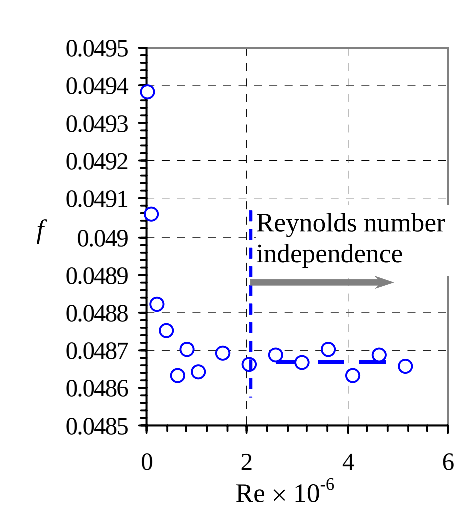



**FIGURE 1**
Nondimensionalized experimental data from a section of pipe.

Finally, we divide both sides of Eq. 3 by $\rho V^2$ to convert $\Delta P$ into an Euler number,

*Nondimensional relationship*:
$$\text{Eu} = \frac{\Delta P}{\rho V^2} = \frac{4\tau_w L}{\rho V^2 D} = \frac{1}{2}\frac{L}{D}\left(\frac{8\tau_w}{\rho V^2}\right) \qquad (4)$$

We recognize the term in parentheses on the right as the Darcy friction factor. Thus,

*Final nondimensional relationship*:
$$\boxed{\text{Eu} = \frac{1}{2}\frac{L}{D} f} \quad \text{or} \quad \boxed{f = 2\frac{D}{L}\text{Eu}} \qquad (5)$$

(*b*) We use Eq. 5 to calculate $f$ at each data point of Table P7-74. We plot $f$ as a function of Re in Fig. 1. We see that the behavior of $f$ mimics that of Eu (as it must because of Eq. 5 where we see that $f$ is just a constant times Eu). Since Eu shows Reynolds number independence for Re greater than about 2 × 10$^6$, so does $f$. **We see Reynolds number independence for Re greater than about 2 × 10$^6$**. From the plot, the extrapolated value of $f$ at large Re is about 0.04867, which agrees with Eq. 5 when we plug in the Re-independent value of Eu,

*Extrapolated value of f*:
$$f = 2\frac{D}{L}\text{Eu} = 2\frac{0.104\text{ m}}{1.3\text{ m}}(0.3042) = \mathbf{0.0487} \qquad (6)$$

***Discussion*** We show in Chap. 8 (the Moody chart) that $f$ does indeed flatten out at high enough values of Re, depending on the relative roughness height, $\varepsilon/D$.

## 7-89

**Solution** We are to create characteristic scales so that we can define a desired established dimensionless parameter.

***Analysis*** (*a*) For Froude number we need a velocity scale, a length scale, and gravity. We already have a length scale and gravity. We create a velocity scale as $\dot{V}'/L$. We then define a Froude number as

*Froude number*:

$$\mathrm{Fr} = \frac{V}{\sqrt{gL}} = \frac{\dot{V}'}{L\sqrt{gL}} = \frac{\dot{V}'}{\sqrt{gL^3}}$$

(*b*) For Reynolds number we need a velocity scale, a length scale, and kinematic viscosity. Of these we only have the kinematic viscosity, so we need to create a velocity scale and a length scale. After a "back of the envelope" analysis, we create a velocity scale as $\dot{V}'/L$ where $L$ is some undefined characteristic length scale. Thus,

*Reynolds number*:

$$\mathrm{Re} = \frac{LV}{\nu} = \frac{L\dot{V}'}{L\nu} = \frac{\dot{V}'}{\nu}$$

Note that in this case, the length scales drop out, so it doesn't matter that we could not define a length scale from the given parameters.

(*c*) For Richardson number we need a length scale, the gravitational constant, a volume flow rate, a density, and a density difference. Of these we have all but the volume flow rate, so we create a volume flow rate scale as $\dot{V}'\,L$. Thus,

*Richardson number*:

$$\mathrm{Ri} = \frac{L^5 g\Delta\rho}{\rho\dot{V}^2} = \frac{L^5 g\Delta\rho}{\rho\left(\dot{V}'L\right)^2} = \frac{L^3 g\Delta\rho}{\rho\left(\dot{V}'\right)^2}$$

***Discussion*** You can verify that each of the parameters above is dimensionless.

**7-90**

**Solution** We are to find the functional relationship between the given parameters and name any established dimensionless parameters.

***Assumptions*** **1** The given parameters are the only ones relevant to the flow at hand.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are seven parameters in this problem; $n = 7$,

*List of relevant parameters*: $$V = f\left(d, D, \rho, \mu, h, g\right) \qquad n = 7 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $V$ | $d$ | $D$ | $\rho$ | $\mu$ | $h$ | $g$ |
|---|---|---|---|---|---|---|
| $\left\{L^1t^{-1}\right\}$ | $\left\{L^1\right\}$ | $\left\{L^1\right\}$ | $\left\{m^1L^{-3}\right\}$ | $\left\{m^1L^{-1}t^{-1}\right\}$ | $\left\{L^1\right\}$ | $\left\{L^1t^{-2}\right\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected Πs*: $$k = n - j = 7 - 3 = 4$$

**Step 4** We need to choose three repeating parameters since $j = 3$. We pick length scale $h$, fluid density $\rho$, and gravitational constant $g$.

*Repeating parameters*: $h$, $\rho$, and $g$

**Step 5** The Πs are generated. Note that in this case we do the algebra in our heads since these relationships are very simple. The dependent Π is

$\Pi_1$ = *a Froude number*: $$\Pi_1 = \frac{V}{\sqrt{gh}}$$

This Π is a type of **Froude number**. Similarly, the two length-scale Πs are obtained easily,

$\Pi_2$: $$\Pi_2 = \frac{d}{h}$$

and

$\Pi_3$: $$\Pi_3 = \frac{D}{h}$$

Finally, the Π formed with viscosity is generated,

$$\Pi_4 = \mu h^{a_4}\rho^{b_4}g^{c_4} \qquad \left\{\Pi_4\right\} = \left\{\left(m^1L^{-1}t^{-1}\right)\left(L^1\right)^{a_4}\left(m^1L^{-3}\right)^{b_4}\left(L^1t^{-2}\right)^{c_4}\right\}$$

*mass:* $$\left\{m^0\right\} = \left\{m^1m^{b_4}\right\} \qquad 0 = 1 + b_4 \qquad b_4 = -1$$

*time:* $$\left\{t^0\right\} = \left\{t^{-1}t^{-2c_4}\right\} \qquad 0 = -1 - 2c_4 \qquad c_4 = -\frac{1}{2}$$

*length:* $\left\{\mathrm{L}^0\right\} = \left\{\mathrm{L}^{-1}\mathrm{L}^{a_4}\mathrm{L}^{-3b_4}\mathrm{L}^{c_4}\right\}$ $\quad$ $\begin{aligned} 0 &= -1 + a_4 - 3b_4 + c_4 \\ 0 &= -1 + a_4 + 3 - \tfrac{1}{2} \end{aligned}$ $\quad$ $a_4 = -\dfrac{3}{2}$

which yields

$\Pi_4$: $$\Pi_4 = \frac{\mu}{\rho h^{\frac{3}{2}}\sqrt{g}}$$

We recognize this $\Pi$ as the inverse of a kind of **Reynolds number**. We also split the *h* terms to separate them into a length scale and (when combined with *g*) a velocity scale. The final form is

*Modified* $\Pi_4$ *= a Reynolds number*: $$\Pi_4 = \frac{\rho h\sqrt{gh}}{\mu}$$

**Step 6** We write the final functional relationship as

*Relationship between* $\Pi$*s*: $$\boxed{\frac{V}{\sqrt{gh}} = f\left(\frac{d}{h}, \frac{D}{h}, \frac{\rho h\sqrt{gh}}{\mu}\right)} \qquad (2)$$

***Discussion*** You may choose different repeating variables, and may generate different nondimensional groups. If you do the algebra correctly, your answer is not "wrong" – you just may not get the same dimensionless groups.

**7-91**

**Solution** We are to find a dimensionless relationship among the given parameters.

***Assumptions*** **1** The given parameters are the only ones relevant to the flow at hand.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are seven parameters in this problem; $n = 7$,

*List of relevant parameters*: $$t_{\text{empty}} = f\left(d, D, \rho, \mu, h, g\right) \qquad n = 7 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $t_{\text{empty}}$ | $d$ | $D$ | $\rho$ | $\mu$ | $h$ | $g$ |
|---|---|---|---|---|---|---|
| $\{t^1\}$ | $\{L^1\}$ | $\{L^1\}$ | $\{m^1L^{-3}\}$ | $\{m^1L^{-1}t^{-1}\}$ | $\{L^1\}$ | $\{L^1t^{-2}\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected Πs*: $$k = n - j = 7 - 3 = 4$$

**Step 4** We need to choose three repeating parameters since $j = 3$. We pick length scale $h$, fluid density $\rho$, and gravitational constant $g$. (Note: these are the same repeating parameters as in the previous problem.)

*Repeating parameters*: $h$, $\rho$, and $g$

**Step 5** The Πs are generated. We leave out the details since the algebra is trivial and can be done by inspection in most cases. The dependent Π is

$\Pi_1$: $$\Pi_1 = t_{\text{empty}}\sqrt{\frac{g}{h}}$$

The rest of the Πs are identical to those of the previous problem.

**Step 6** We write the final functional relationship as

*Relationship between Πs*: $$\boxed{t_{\text{empty}}\sqrt{\frac{g}{h}} = f\left(\frac{d}{h}, \frac{D}{h}, \frac{\rho h\sqrt{gh}}{\mu}\right)} \qquad (2)$$

***Discussion*** You may choose different repeating variables, and may generate different nondimensional groups. If you do the algebra correctly, your answer is not "wrong" – you just may not get the same dimensionless groups.

**7-92**

**Solution** We are to calculate the temperature of water in a model test to ensure similarity with the prototype, and we are to predict the time required to empty the prototype tank.

***Assumptions*** **1** The parameters specified in the previous problem are the only parameters relevant to the problem. **2** The model and prototype are geometrically similar.

***Properties*** For ethylene glycol at 60°C, $\nu = \mu/\rho = 4.75\times10^{-6}$ m²/s (given).

***Analysis***

(*a*) We use the functional relationship obtained in the previous problem,

*Dimensionless relationship*:
$$t_{\text{empty}}\sqrt{\frac{g}{h}} = f\left(\frac{d}{h},\frac{D}{h},\frac{\rho h\sqrt{gh}}{\mu}\right) \quad (1)$$

Since the model and prototype are geometrically similar, $(d/h)_{\text{model}} = (d/h)_{\text{prototype}}$ and $(D/h)_{\text{model}} = (D/h)_{\text{prototype}}$. Thus, we are left with only one Π to match to ensure similarity. Namely, the Reynolds number parameter in Eq. 1 must be matched between model and prototype. Since $g$ remains the same in either case, and using "m" for model and "p" for prototype,

*Similarity*:
$$\left(\frac{\rho h\sqrt{gh}}{\mu}\right)_{\text{m}} = \left(\frac{\rho h\sqrt{gh}}{\mu}\right)_{\text{p}} \quad \text{or} \quad \frac{\rho_{\text{m}}}{\mu_{\text{m}}} = \frac{\rho_{\text{p}}}{\mu_{\text{p}}}\left(\frac{h_{\text{p}}}{h_{\text{m}}}\right)^{\frac{3}{2}} \quad (2)$$

We recognize that $\nu = \mu/\rho$, and we know that $h_p/h_m = 4$. Thus, Eq. 2 reduces to

*Similarity*:
$$\nu_{\text{m}} = \nu_{\text{p}}\left(\frac{h_{\text{p}}}{h_{\text{m}}}\right)^{\frac{-3}{2}} = 4.75\times10^{-6}\ \text{m}^2/\text{s}\left(4\right)^{\frac{-3}{2}} = 5.94\times10^{-7}\ \text{m}^2/\text{s} \quad (3)$$

For similarity we need to find the temperature of water where the kinematic viscosity is $5.94\times10^{-7}$ m²/s. By interpolation from the property tables, **the designers should run the model tests at a water temperature of 45.8°C**.

(*b*) At dynamically similar conditions, Eq. 1 yields

*At dynamically similar conditions*:

$$\left(t_{\text{empty}}\sqrt{\frac{g}{h}}\right)_{\text{p}} = \left(t_{\text{empty}}\sqrt{\frac{g}{h}}\right)_{\text{m}} \quad \rightarrow \quad t_{\text{empty,p}} = t_{\text{empty,m}}\sqrt{\frac{h_{\text{p}}}{h_{\text{m}}}} = 4.53\ \text{min}\sqrt{4} = \mathbf{9.06\ min} \quad (5)$$

***Discussion*** We set up Eqs. 3 and 5 in terms of ratios of $h_p$ to $h_m$ so that the actual dimensions are not needed – just the *ratio* is needed, and it is given.

**7-93**

**Solution** For the simplified case in which *V* depends only on *h* and *g*, we are to determine how *V* increases when *h* is doubled.

***Assumptions*** **1** The given parameters are the only ones relevant to the problem.

***Analysis*** We employ the dimensional analysis results of Problem 7-90. Dropping *d*, *D*, $\rho$, and $\mu$ from the list of parameters, we are left with $n = 3$,

*List of relevant parameters*: $$V = f(h, g) \qquad n = 3 \qquad (1)$$

We perform the analysis in our heads –only one Π remains, and it is therefore set to a constant. The final result of the dimensional analysis is

*Relationship between Πs*: $$\frac{V}{\sqrt{gh}} = \text{constant} \qquad (2)$$

Thus, when *h* is doubled, we can easily calculate the factor by which V increases,

*Increase in V*: $$\boxed{\frac{V_2}{\sqrt{gh_2}} = \frac{V_1}{\sqrt{gh_1}} \quad \text{or} \quad V_2 = V_1\sqrt{\frac{h_2}{h_1}} = V_1\sqrt{2}} \qquad (3)$$

Thus, **when *h* increases by a factor of 2, *V* increases by a factor of** $\sqrt{2}$ .

***Discussion*** We don't need to know the constant in Eq. 2 to solve the problem. However, it turns out that the constant is $\sqrt{2}$ (see Chap. 5).

**7-94**

**Solution** We are to verify the dimensions of particle relaxation time $\tau_p$, and then identify the established dimensionless parameter formed by nondimensionalization of $\tau_p$.

***Analysis*** First we obtain the primary dimensions of $\tau_p$,

*Primary dimensions of* $\tau_p$: $$\{\tau_p\} = \left\{ \frac{\frac{\text{m}}{\text{L}^3} \times \text{L}^2}{\frac{\text{m}}{\text{Lt}}} \right\} = \{\mathbf{t}\}$$

A characteristic time scale for the air flow is *L*/*V*. Thus, we nondimensionalize $\tau_p$,

*Nondimensionalized particle relaxation time*: $$\tau_p^* = \tau_p = \frac{\rho_p d_p^{\ 2}}{18\mu}\frac{V}{L}$$

From Table 7-5 we recognize this as the **Stokes number**, Stk,

*Stokes number*: $$\boxed{\text{Stk} = \frac{\rho_p d_p^{\ 2} V}{18\mu L}}$$

***Discussion*** Stokes number is useful when studying the flow of aerosol particles.

**7-95**

**Solution** We are to compare the primary dimensions of each given property in mass-based and force-based primary dimensions, and discuss.

***Analysis*** From previous problems and examples in this chapter, we can write down the primary dimensions of each property in the mass-based system. We use the fundamental definitions of these quantities to generate the primary dimensions in the force-based system:

(*a*) For pressure *P* the primary dimensions are

| **Mass-based primary dimensions** | **Force-based primary dimensions** |
| --- | --- |
| $\{P\} = \left\{\frac{\text{m}}{\text{t}^2\text{L}}\right\}$ | $\{P\} = \left\{\frac{\text{force}}{\text{area}}\right\} = \left\{\frac{\text{F}}{\text{L}^2}\right\}$ |

(*b*) For moment $\vec{M}$ the primary dimensions are

| **Mass-based primary dimensions** | **Force-based primary dimensions** |
| --- | --- |
| $\{\vec{M}\} = \left\{\text{m}\frac{\text{L}^2}{\text{t}^2}\right\}$ | $\{\vec{M}\} = \{\text{force} \times \text{moment arm}\} = \{\text{FL}\}$ |

(*c*) For energy *E* the primary dimensions are

| **Mass-based primary dimensions** | **Force-based primary dimensions** |
| --- | --- |
| $\{E\} = \left\{\text{m}\frac{\text{L}^2}{\text{t}^2}\right\}$ | $\{E\} = \{\text{force} \times \text{distance}\} = \{\text{FL}\}$ |

We see that (in these three examples anyway), the forced-base cases have only two primary dimensions represented (F and L), whereas the mass-based cases have three primary dimensions represented (m, L, and t). Some authors would prefer the force-based system because of its **reduced complexity** when dealing with forces, pressures, energies, etc.

***Discussion*** Not all variables have a simpler form in the force-based system. Mass itself for example has primary dimensions of {m} in the mass-based system, but has primary dimensions of $\{\text{Ft}^2/\text{L}\}$ in the force-based system. In problems involving mass, mass flow rates, and/or density, the force-based system may not have any advantage.

**7-96**

**Solution** The pressure difference between the inside of a soap bubble and the outside air is to be analyzed with dimensional analysis and the method of repeating variables using the force-based system of primary dimensions.

***Assumptions*** **1** The soap bubble is neutrally buoyant in the air, and gravity is not relevant. **2** No other variables or constants are important in this problem.

***Analysis*** The step-by-step method of repeating variables is employed.

**Step 1** There are three variables and constants in this problem; $n$ = 3,

$$\Delta P = f\left(R, \sigma_s\right) \qquad n = 3 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed. The dimensions of pressure are force per area and those of surface tension are force per length.

| $\Delta P$ | $R$ | $\sigma_s$ |
|---|---|---|
| $\{F^1L^{-2}\}$ | $\{L^1\}$ | $\{F^1L^{-1}\}$ |

**Step 3** As a first guess, $j$ is set equal to 2, the number of primary dimensions represented in the problem (F and L).

*Reduction*: $j = 2$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $k = n - j = 3 - 2 = 1$

**Step 4** We choose two repeating parameters since $j$ = 2. Our only choice is $R$ and $\sigma_s$ since $\Delta P$ is the dependent variable.

**Step 5** The dependent Π is generated:

$$\Pi_1 = \Delta P R^{a_1} \sigma_s^{b_1} \qquad \{\Pi_1\} = \{F^0L^0\} = \left\{\left(F^1L^{-2}\right)L^{a_1}\left(F^1L^{-1}\right)^{b_1}\right\}$$

*force*: $\{F^0\} = \{F^1F^{b_1}\}$ $\qquad 0 = 1 + b_1 \qquad b_1 = -1$

*length*: $\{L^0\} = \{L^{-2}L^{a_1}L^{-b_1}\}$ $\qquad 0 = -2 + a_1 - b_1$, $a_1 = 2 + b_1 \qquad a_1 = 1$

Eq. 1 thus becomes

$\Pi_1$:
$$\Pi_1 = \frac{\Delta P R}{\sigma_s} \qquad (2)$$

From Table 7-5, the established nondimensional parameter most similar to Eq. 2 is the **Weber number**, defined as a pressure times a length divided by surface tension. There is no need to further manipulate this Π.

**Step 6** We now write the functional relationship between the nondimensional parameters. Since there is only one Π, it is a function of *nothing*, which means it must be a constant,

*Relationship between* Π*s*:
$$\Pi_1 = \frac{\Delta P R}{\sigma_s} = f\left(\text{nothing}\right) = \text{constant} \quad \rightarrow \quad \boxed{\Delta P = \text{constant}\frac{\sigma_s}{R}} \qquad (3)$$

The result using force-based primary dimensions is indeed identical to the previous result using the mass-based system.

***Discussion*** Because only two primary dimensions are represented in the problem when using the force-based system, the algebra is in fact a lot easier.

**7-97**

**Solution** We are to a third established nondimensional parameter that is formed by the product or ratio of two given established nondimensional parameters.

***Analysis***

(*a*) The product of Reynolds number and Prandtl number yields

*Reynolds number times Prandtl number*: $$\text{Re}\times\text{Pr}=\frac{\rho LV}{\mu}\times\frac{c_P\mu}{k}=\frac{\rho LVc_P}{k} \quad (1)$$

We recognize Eq. 1 as the *Peclet number*,

*Peclet number*: $$\boxed{\text{Pe}=\text{Re}\times\text{Pr}=\frac{\rho LVc_P}{k}=\frac{LV}{\alpha}} \quad (2)$$

(*b*) The ratio of Schmidt number and Prandtl number yields

*Schmidt number divided by Prandtl number*: $$\frac{\text{Sc}}{\text{Pr}}=\frac{\dfrac{\mu}{\rho D_{AB}}}{\dfrac{c_P\mu}{k}}=\frac{k}{\rho c_P D_{AB}} \quad (3)$$

We recognize Eq. 3 as the *Lewis number*,

*Lewis number*: $$\boxed{\text{Le}=\frac{\text{Sc}}{\text{Pr}}=\frac{k}{\rho c_P D_{AB}}=\frac{\alpha}{D_{AB}}} \quad (4)$$

(*c*) The product of Reynolds number and Schmidt number yields

*Reynolds number times Schmidt number*: $$\text{Re}\times\text{Sc}=\frac{\rho LV}{\mu}\times\frac{\mu}{\rho D_{AB}}=\frac{LV}{D_{AB}} \quad (5)$$

We recognize Eq. 5 as the *Sherwood number*,

*Sherwood number*: $$\boxed{\text{Sh}=\text{Re}\times\text{Sc}=\frac{LV}{D_{AB}}} \quad (6)$$

***Discussion*** Can you find any other such combinations from Table 7-5?

**7-98**

**Solution** We are to determine the relationship between four established nondimensional parameters, and then try to form the Stanton number by some combination of only *two* other established dimensionless parameters.

***Analysis*** We manipulate Re, Nu, and Pr, guided by the known result. After some trial and error,

*Stanton number*:

$$\mathrm{St} = \frac{\mathrm{Nu}}{\mathrm{Re}\times\mathrm{Pr}} = \frac{\dfrac{Lh}{k}}{\dfrac{\rho VL}{\mu}\times\dfrac{\mu c_P}{k}} = \frac{h}{\rho c_p V} \qquad (1)$$

We recognize from Table 7-5 (or from the previous problem) that *Peclet number* is equal to the product of Reynolds number and Prandtl number. Thus,

*Stanton number*:

$$\mathrm{St} = \frac{\mathrm{Nu}}{\mathrm{Pe}} = \frac{\dfrac{Lh}{k}}{\dfrac{\rho LVc_P}{k}} = \frac{h}{\rho c_p V} \qquad (2)$$

***Discussion*** Not all named, established dimensionless parameters are independent of other named, established dimensionless parameters.

---

**7-99**

**Solution** We are to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** First we do some thinking. If we imagine traveling at the same speed as the bottom plate, the flow would be identical to that of Problem 7-56 except that the top plate speed would be ($V_{\mathrm{top}} - V_{\mathrm{bottom}}$) instead of just $V$. The step-by-step method of repeating variables is otherwise identical to that of Problem 7-56, and the details are not included here. The final functional relationship is

*Relationship between Πs*:

$$\frac{u}{V_{\mathrm{top}} - V_{\mathrm{bottom}}} = f\left(\mathrm{Re}, \frac{y}{h}\right) \qquad (1)$$

where

*Reynolds number*:

$$\mathrm{Re} = \frac{\rho\left(V_{\mathrm{top}} - V_{\mathrm{bottom}}\right)h}{\mu} \qquad (2)$$

***Discussion*** It is always wise to look for shortcuts like this to save us time.

---

**7-100**

**Solution** We are to determine the primary dimensions of electrical charge.

***Analysis*** The fundamental definition of electrical current is charge per unit time. Thus,

*Primary dimensions of charge*:

$$\{q\} = \{\text{current}\times\text{time}\} = \{\mathbf{I\ t}\} \qquad (1)$$

Or, in exponent form, $\{q\} = \{\mathbf{t}^1\,\mathbf{I}^1\}$.

***Discussion*** We see that all dimensions, even those of electrical properties, can be expressed in terms of primary dimensions.

---

**7-101**

**Solution** We are to determine the primary dimensions of electrical capacitance.

***Analysis*** Electrical capacitance $C$ is measured in units of farads (F). By definition, a one-farad capacitor with an applied electric potential of one volt across it will store one coulomb of electrical charge. Thus,

*Primary dimensions of capacitance*:

$$\{C\} = \{\text{charge / voltage}\} = \left\{ \frac{\text{I t}}{\frac{\text{mL}^2}{\text{t}^3\text{I}}} \right\} = \left\{ \frac{\mathbf{I^2 t^4}}{\mathbf{mL^2}} \right\} \qquad (1)$$

where the primary dimensions of voltage are obtained from Problem 7-10, and those of electric charge are obtained from the previous problem. Or, in exponent form, $\{C\}$ = $\{\mathbf{m^{-1} L^{-2} t^4 I^2}\}$.

***Discussion*** We see that all dimensions, even those of electrical properties, can be expressed in terms of primary dimensions.

---

**7-102**

**Solution** We are to determine the primary dimensions of electrical time constant *RC*, and discuss the significance of our result.

***Analysis*** The primary dimensions of electrical resistance are obtained from Problem 7-11. Those of electrical capacitance $C$ are obtained from the previous problem. Thus,

*Primary dimensions of electrical time constant RC*:

$$\{RC\} = \{\text{resistance} \times \text{capacitance}\} = \left\{ \frac{\text{mL}^2}{\text{t}^3\text{I}^2} \times \frac{\text{I}^2\text{t}^4}{\text{mL}^2} \right\} = \{\mathbf{t}\} \qquad (1)$$

Thus we see that the primary dimensions of *RC* are those of *time*. This explains why a resistor and capacitor in series is often used in timing circuits.

***Discussion*** The cut-off frequency of the low-pass filter is proportional to $1/R\cdot C$. If the resistor and the capacitor were to swap places we would have a high-pass rather than a low-pass filter.

---

**7-103**

**Solution** We are to determine the primary dimensions of both sides of the equation, and we are to verify that the equation is dimensionally homogeneous.

***Analysis*** The primary dimensions of the time derivative ($d/dt$) are 1/time. The primary dimensions of capacitance are current$^2$×time$^4$ / (mass×length$^2$), as obtained from Problem 7-101. Thus both sides of the equation can be written in terms of primary dimensions,

$$\{I\} = \{\text{current}\} \qquad\qquad \{I\} = \{\text{I}\}$$

$$C\frac{dE}{dt} = \left\{ \frac{\text{current}^2 \times \text{time}^4}{\text{mass} \times \text{length}^2} \frac{\frac{\text{mass} \times \text{length}^2}{\text{current} \times \text{time}^3}}{\text{time}} \right\} = \{\text{current}\} \qquad\qquad \left\{ C\frac{dE}{dt} \right\} = \{\text{I}\}$$

Indeed, **both sides of the equation have the same dimensions, namely {*I*}**.

***Discussion*** Current is one of our seven primary dimensions. These results verify our algebra in Problem 7-101.

---

**7-104**

**Solution** We are to find the functional relationship between the given parameters, and then answer some questions about scaling.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis***

(*a*) The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are four parameters in this problem; $n = 4$,

*List of relevant parameters*: $$\delta P = f\left(\rho, \dot{V}, D\right) \qquad n = 4 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $\delta P$ | $\rho$ | $\dot{V}$ | $D$ |
|---|---|---|---|
| $\{m^1 L^{-1} t^{-2}\}$ | $\{m^1 L^{-3}\}$ | $\{L^3 t^{-1}\}$ | $\{L^1\}$ |

**Step 3** As a first guess, *j* is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of *j* is correct, the expected number of Πs is

*Number of expected Πs*: $$k = n - j = 4 - 3 = 1$$

**Step 4** We need to choose three repeating parameters since $j = 3$. We only have one choice in this problem, since there are only three independent parameters on the right-hand side of Eq. 1,

*Repeating parameters*: $\rho$, $\dot{V}$, and $D$

**Step 5** The dependent Π is generated:

$$\Pi_1 = \delta P \rho^{a_1} \dot{V}^{b_1} D^{c_1} \qquad \{\Pi_1\} = \left\{\left(m^1 L^{-1} t^{-2}\right)\left(m^1 L^{-3}\right)^{a_1}\left(L^3 t^{-1}\right)^{b_1}\left(L^1\right)^{c_1}\right\}$$

*mass*: $\{m^0\} = \{m^{1+a_1}\}$ $\qquad 0 = 1 + a_1 \qquad a_1 = -1$

*time*: $\{t^0\} = \{t^{-2-b_1}\}$ $\qquad 0 = -2 - b_1 \qquad b_1 = -2$

*length*: $\{L^0\} = \{L^{-1} L^{-3a_1} L^{3b_1} L^{c_1}\}$ $\qquad 0 = -1 - 3a_1 + 3b_1 + c_1 \qquad c_1 = 4$

The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{D^4 \delta P}{\rho \dot{V}^2}$$

**Step 6** Since there is only one Π, it is a function of nothing. This is only possible if we set the Π equal to a constant. We write the final functional relationship as

*Relationship between Πs*: $$\boxed{\Pi_1 = \frac{D^4 \delta P}{\rho \dot{V}^2} = \text{constant}} \qquad (2)$$

(*b*) We re-write Eq. 2 as

*Equation for δP*: $$\delta P = \text{constant}\frac{\rho \dot{V}^2}{D^4} \qquad (3)$$

Thus, **if we double the size of the cyclone, the pressure drop will *decrease* by a factor of $2^4$ = 16**.

(c) Also from Eq. 3 we see that **if we double the volume flow rate, the pressure drop will increase by a factor of $2^2$ = 4**.

***Discussion*** The pressure drop would be smallest for the *largest* cyclone operating at the *smallest* volume flow rate. (This agrees with our intuition.)

**7-105**

**Solution** We are to find the functional relationship between the given parameters, and then answer some questions about scaling.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis***

(*a*) The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are five parameters in this problem; $n = 5$,

*List of relevant parameters*: $$w = f\left(q_p, E_f, \mu, D_p\right) \qquad n = 5 \qquad (1)$$

**Step 2** The primary dimensions of each parameter are listed,

| $w$ | $q_p$ | $E_f$ | $\mu$ | $D_p$ |
|---|---|---|---|---|
| $\{L^1t^{-1}\}$ | $\{I^1t^1\}$ | $\{m^1L^1t^{-3}I^{-1}\}$ | $\{m^1L^{-1}t^{-1}\}$ | $\{L^1\}$ |

where the primary dimensions of voltage are obtained from Problem 7-10, and those of electric charge are obtained from Problem 7-100.

**Step 3** As a first guess, $j$ is set equal to 4, the number of primary dimensions represented in the problem (m, L, t, and I).

*Reduction*: $$j = 4$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 5 - 4 = 1$$

**Step 4** We need to choose four repeating parameters since $j = 4$. We only have one choice in this problem, since there are only four independent parameters on the right-hand side of Eq. 1,

*Repeating parameters*: $q_p$, $E$, $D_p$, and $\mu$

**Step 5** The dependent Π is generated:

$$\Pi_1 = wq_p^{a_1}E_f^{b_1}\mu^{c_1}D_p^{d_1} \qquad \{\Pi_1\} = \left\{\left(L^1t^{-1}\right)\left(I^1t^1\right)^{a_1}\left(m^1L^1t^{-3}I^{-1}\right)^{b_1}\left(m^1L^{-1}t^{-1}\right)^{c_1}\left(L^1\right)^{d_1}\right\}$$

*current*: $$\{I^0\} = \{I^{a_1}I^{-b_1}\} \qquad 0 = a_1 - b_1 \qquad a_1 = b_1$$

*mass*: $$\{m^0\} = \{m^{b_1}m^{c_1}\} \qquad 0 = b_1 + c_1 \qquad c_1 = -b_1 = -a_1$$

*time*: $$\{t^0\} = \{t^{-1}t^{a_1}t^{-3b_1}t^{-c_1}\} \qquad 0 = -1 + a_1 - 3b_1 - c_1 \qquad a_1 = b_1 = -1, \; c_1 = 1$$

*length*: $$\{L^0\} = \{L^1L^{b_1}L^{-c_1}L^{d_1}\} \qquad 0 = 1 + b_1 - c_1 + d_1 \qquad d_1 = 1$$

The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{w\mu D_p}{q_pE_f}$$

**Step 6** Since there is only one Π, it is a function of nothing. This is only possible if we set the Π equal to a constant. We write the final functional relationship as

*Relationship between Πs*:

$$\Pi_1 = \frac{w\mu D_p}{q_p E} = \text{constant} \qquad (2)$$

(*b*) We re-write Eq. 2 as

*Equation for w*:

$$w = \text{constant}\,\frac{q_p E_f}{\mu D_p} \qquad (3)$$

Thus, **if we double the electric field strength, the drift velocity will increase by a factor of 2**.

(c) Also from Eq. 3 we see that **if we double the particle size, the drift velocity will decrease by a factor of 2**.

***Discussion*** These results agree with our intuition. Certainly we would expect the drift velocity to increase if we increase the field strength. Also, larger particles have more aerodynamic drag, so for the same charge, we would expect a larger dust particle to drift more slowly than a smaller dust particle.

**7-106**

**Solution** We are to generate a dimensionless functional relationship between the given parameters and then compare our results with a known exact analytical solution.

***Assumptions*** **1** There is no flow (hydrostatics). **2** The parameters listed here are the only relevant parameters in the problem.

***Analysis*** (*a*) We perform a dimensional analysis using the method of repeating variables.

**Step 1** There are five parameters in this problem; $n = 6$,

*List of relevant parameters*: $h = f(\rho, g, \sigma_s, D, \phi) \qquad n = 6 \qquad (1)$

**Step 2** The primary dimensions of each parameter are listed,

| $h$ | $\rho$ | $g$ | $\sigma_s$ | $D$ | $\phi$ |
|---|---|---|---|---|---|
| $\{L^1\}$ | $\{m^1L^{-3}\}$ | $\{L^1t^{-2}\}$ | $\{m^1t^{-2}\}$ | $\{L^1\}$ | $\{1\}$ |

Note that the dimensions of the contact angle are unity (angles are dimensionless).

**Step 3** As a first guess, *j* is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $j = 3$

If this value of *j* is correct, the expected number of Πs is

*Number of expected* Π*s*: $k = n - j = 6 - 3 = 3$

**Step 4** We need to choose three repeating parameters since $j = 3$. We cannot choose $\phi$ since it is dimensionless. We choose a length ($D$) and a density ($\rho$). We'd rather have gravitational constant $g$ than surface tension $\sigma_s$ in our Πs. So, we choose

*Repeating parameters*: $\rho$, $g$, $D$

**Step 5** The dependent Π is generated. Since *h* has the same dimensions as *D*, we immediately write

$\Pi_1$: $$\Pi_1 = \frac{h}{D}$$

The first independent Π is generated by combining $\sigma_s$ with the repeating parameters,

$$\Pi_2 = \sigma_s \rho^{a_2} g^{b_2} D^{c_2} \qquad \{\Pi_2\} = \left\{(m^1t^{-2})(m^1L^{-3})^{a_2}(L^1t^{-2})^{b_2}(L^1)^{c_2}\right\}$$

*mass*: $\{m^0\} = \{m^1m^{a_2}\}$ $\qquad 0 = 1 + a_2 \qquad a_2 = -1$

*time*: $\{t^0\} = \{t^{-2}t^{-2b_2}\}$ $\qquad 0 = -2 - 2b_2 \qquad b_2 = -1$

*length*: $\{L^0\} = \{L^{-3a_2}L^{b_2}L^{c_2}\}$ $\qquad 0 = -3a_2 + b_2 + c_2$, $\; c_2 = 3a_2 - b_2 \qquad c_2 = -2$

The first independent Π is thus

$\Pi_2$: $$\Pi_2 = \frac{\sigma_s}{\rho g D^2}$$

Finally, the third Π (second independent Π) is simply angle $\phi$ itself since it is dimensionless,

$\Pi_3$: $$\Pi_3 = \phi$$

**Step 6** We write the final functional relationship as

*Relationship between* $\Pi$*s*: $$\boxed{\frac{h}{D} = f\left(\frac{\sigma_s}{\rho g D^2}, \phi\right)} \qquad (2)$$

(*b*) From Chap. 2 we see that the exact analytical solution is

*Exact relationship*: $$h = \frac{4\sigma_s}{\rho g D}\cos\phi \qquad (3)$$

Comparing Eqs. 2 and 3, we see that they are indeed of the same form. In fact,

*Functional relationship*: $$\boxed{\Pi_1 = \text{constant} \times \Pi_2 \times \cos\Pi_3} \qquad (4)$$

***Discussion*** We cannot determine the constant in Eq. 4 by dimensional analysis. However, one experiment is enough to establish the constant. Or, in this case we can find the constant exactly. Viscosity is not relevant in this problem since there is no fluid motion.

---

## 7-107

**Solution** We are to find a functional relationship for the time scale required for the liquid to climb up the capillary tube.

***Assumptions*** **1** $t_{\text{rise}}$ is a function of the same parameters listed in the previous problem, but there is another relevant parameter.

***Analysis*** Since this is an unsteady problem, the rise time will surely depend also on fluid viscosity $\mu$. The list of parameters now involves seven parameters,

*List of relevant parameters*: $$t_{\text{rise}} = f\left(\rho, g, \sigma_s, D, \phi, \mu\right) \qquad n = 7 \qquad (1)$$

and we expect four $\Pi$s. We choose the same repeating parameters and the algebra is similar to that of the previous problem. It turns out that

$\Pi_1$: $$\Pi_1 = t_{\text{rise}}\sqrt{\frac{g}{D}}$$

The second and third $\Pi$ are the same as those of the previous problem. Finally, the fourth $\Pi$ is formed by combining $\mu$ with the repeating parameters. We expect some kind of Reynolds number. We can do the algebra in our head. Specifically, a velocity scale can be formed as $\sqrt{gD}$ . Thus,

$\Pi_4$: $$\Pi_4 = \text{Re} = \frac{\rho D\sqrt{gD}}{\mu}$$

The final functional relationship is

*Relationship between* $\Pi$*s*: $$\boxed{t_{\text{rise}}\sqrt{\frac{g}{D}} = f\left(\frac{\sigma_s}{\rho g D^2}, \phi, \text{Re}\right)} \qquad (2)$$

***Discussion*** If we would have defined a time scale as $\sqrt{D/g}$ , we could have written $\Pi_1$ by inspection as well, saving ourselves some algebra.

---

**7-108**

**Solution** We are to use dimensional analysis to find the functional relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are four parameters in this problem; $n = 4$,

*List of relevant parameters*:
$$I = f\left(P, c, \rho\right) \qquad n = 4 \qquad (1)$$

**Step 2** The dimensions of $I$ are those of power per area. The primary dimensions of each parameter are listed,

| $I$ | $P$ | $c$ | $\rho$ |
|---|---|---|---|
| $\left\{\text{m}^1\text{t}^{-3}\right\}$ | $\left\{\text{m}^1\text{L}^{-1}\text{t}^{-2}\right\}$ | $\left\{\text{L}^1\text{t}^{-1}\right\}$ | $\left\{\text{m}^1\text{L}^{-3}\right\}$ |

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*:
$$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Π*s*:
$$k = n - j = 4 - 3 = 1$$

**Step 4** We need to choose three repeating parameters since $j = 3$. *The problem is that the three independent parameters form a* Π *all by themselves* ($c^2\rho/P$ is dimensionless). Let's see what happens if we don't notice this, and we pick all three independent parameters as repeating variables,

*Repeating parameters*: $P$, $\rho$, and $c$

**Step 5** The Π is generated:

$$\Pi_1 = I \times P^a \rho^b c^c \qquad \left\{\Pi_1\right\} = \left\{\left(\text{m}^1\text{t}^{-3}\right)\left(\text{m}^1\text{L}^{-1}\text{t}^{-2}\right)^a\left(\text{m}^1\text{L}^{-3}\right)^b\left(\text{L}^1\text{t}^{-1}\right)^c\right\}$$

*mass:*
$$\left\{\text{m}^0\right\} = \left\{\text{m}^1\text{m}^a\text{m}^b\right\} \qquad 0 = 1 + a + b \qquad a = -1 - b$$

*time:*
$$\left\{\text{t}^0\right\} = \left\{\text{t}^{-3}\text{t}^{-2a}\text{t}^{-c}\right\} \qquad \begin{array}{l} 0 = -3 - 2a - c \\ c = -3 - 2a \end{array} \qquad c = -1 + 2b$$

*length:*
$$\left\{\text{L}^0\right\} = \left\{\text{L}^{-a}\text{L}^{-3b}\text{L}^c\right\} \qquad \begin{array}{l} 0 = -a - 3b + c \\ c = a + 3b \end{array} \qquad c = -1 + 2b$$

This is a situation in which two of the equations agree, but we cannot solve for unique exponents. If we knew $b$, we could get $a$ and $c$. The problem is that any value of $b$ we choose will make the Π dimensionless. For example, if we choose $b = 1$, we find that $a = -2$ and $c = 1$, yielding

$\Pi_1$ for the case with $b = 1$:
$$\Pi_1 = \frac{I\rho c}{P^2}$$

Since there is only one Π, we write

Functional relationship for the case with $b = 1$:
$$I = \text{constant} \times \frac{P^2}{\rho c} \qquad (2)$$

However, if we choose a different value of *b*, say *b* = –1, then *a* = 0 and *c* = –3, yielding

$\Pi_1$ for the case with *b* = –1: $$\Pi_1 = \frac{I}{\rho c^3}$$

Since there is only one Π, we write

Functional relationship for the case with *b* = –1: $$I = \text{constant} \times \rho c^3 \qquad (3)$$

Similarly, you can come up with a whole family of possible answers, depending on your choice of *b*. We double check our algebra and realize that any value of *b* works. Hence *the problem is indeterminate with three repeating variables*.

We go back now and realize that something is wrong. As stated previously, the problem is that the three independent parameters can form a dimensionless group all by themselves. This is another case where we have to reduce *j* by 1. Setting *j* = 3 – 1 = 2, we choose two repeating parameters,

*Repeating parameters*: $\rho$ and $c$

We jump to Step 5 of the method of repeating variables,

**Step 5** The first Π is generated:

$$\Pi_1 = I\rho^a c^b \qquad \{\Pi_1\} = \left\{\left(\text{m}^1\text{t}^{-3}\right)\left(\text{m}^1\text{L}^{-3}\right)^a\left(\text{L}^1\text{t}^{-1}\right)^b\right\}$$

*mass:* $\{\text{m}^0\} = \{\text{m}^1\text{m}^a\}$ $\quad 0 = 1 + a \quad$ $a = -1$

*time:* $\{\text{t}^0\} = \{\text{t}^{-3}\text{t}^{-b}\}$ $\quad 0 = -3 - b \quad$ $b = -3$

*length:* $\{\text{L}^0\} = \{\text{L}^{-3a}\text{L}^b\}$ $\quad 0 = -3a + b$, $b = 3a \quad$ $b = -3$

Fortunately, the results for time and length agree. The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{I}{\rho c^3}$$

We form the second Π with sound pressure *P*,

$$\Pi_2 = P\rho^e c^f \qquad \{\Pi_2\} = \left\{\left(\text{m}^1\text{L}^{-1}\text{t}^{-2}\right)\left(\text{m}^1\text{L}^{-3}\right)^e\left(\text{L}^1\text{t}^{-1}\right)^f\right\}$$

*mass:* $\{\text{m}^0\} = \{\text{m}^1\text{m}^e\}$ $\quad 0 = 1 + e \quad$ $e = -1$

*time:* $\{\text{t}^0\} = \{\text{t}^{-2}\text{t}^{-f}\}$ $\quad 0 = -2 - f \quad$ $f = -2$

*length:* $\{\text{L}^0\} = \{\text{L}^{-1}\text{L}^{-3e}\text{L}^f\}$ $\quad 0 = -1 - 3e + f$, $f = 1 + 3e \quad$ $f = -2$

The second Π is thus

$\Pi_2$: $$\Pi_2 = \frac{P}{\rho c^2}$$

**Step 6** We write the final functional relationship as

*Relationship between* Π*s*: $$\boxed{\frac{I}{\rho c^3} = f\left(\frac{P}{\rho c^2}\right)} \qquad (4)$$

(*b*) We try the force-based primary dimension system instead.

**Step 1** There are four parameters in this problem; $n = 4$,

*List of relevant parameters*: $$I = f\left(P, c, \rho\right) \qquad n = 4 \qquad (5)$$

**Step 2** The dimensions of *I* are those of power per area. The primary dimensions of each parameter are listed,

$$\begin{matrix} I & P & c & \rho \\ \left\{F^1L^{-1}t^{-1}\right\} & \left\{F^1L^{-2}\right\} & \left\{L^1t^{-1}\right\} & \left\{F^1t^2L^{-4}\right\} \end{matrix}$$

**Step 3** As a first guess, *j* is set equal to 3, the number of primary dimensions represented in the problem (F, L, and t). Again, however, the three independent parameters form a dimensionless group all by themselves. Thus we lower *j* by 1.

*Reduction*: $$j = 2$$

If this value of *j* is correct, the expected number of Πs is

*Number of expected* Π*s*: $$k = n - j = 4 - 2 = 2$$

**Step 4** We need to choose two repeating parameters since $j = 2$. We pick the same two parameters as in Part (*a*),

*Repeating parameters*: $\rho$ and $c$

**Step 5** The first Π is generated:

$$\Pi_1 = I \times \rho^b c^c \qquad \left\{\Pi_1\right\} = \left\{\left(F^1L^{-1}t^{-1}\right)\left(F^1t^2L^{-4}\right)^b\left(L^1t^{-1}\right)^c\right\}$$

*force*: $$\left\{F^0\right\} = \left\{F^1F^b\right\} \qquad 0 = 1 + b \qquad b = -1$$

*time*: $$\left\{t^0\right\} = \left\{t^{-1}t^{2b}t^{-c}\right\} \qquad \begin{matrix} 0 = -1 + 2b - c \\ c = -1 + 2b \end{matrix} \qquad c = -3$$

*length*: $$\left\{L^0\right\} = \left\{L^{-1}L^{-4b}L^c\right\} \qquad \begin{matrix} 0 = -1 - 4b + c \\ c = 1 + 4b \end{matrix} \qquad c = -3$$

Again the two results for length and time agree. The dependent Π is thus

$\Pi_1$: $$\Pi_1 = \frac{I}{\rho c^3}$$

We form the second Π with sound pressure *P*,

$$\Pi_2 = P \times \rho^e c^f \qquad \left\{\Pi_2\right\} = \left\{\left(F^1L^{-2}\right)\left(F^1t^2L^{-4}\right)^e\left(L^1t^{-1}\right)^f\right\}$$

*force:* $$\left\{F^0\right\} = \left\{F^1F^e\right\} \qquad 0 = 1 + e \qquad e = -1$$

*time:* $\left\{ \text{t}^0 \right\} = \left\{ \text{t}^{-2e} \text{t}^{-f} \right\}$ $\quad 0 = -2e - f \quad f = -2e \quad f = -2$

*length:* $\left\{ \text{L}^0 \right\} = \left\{ \text{L}^{-2} \text{L}^{-4e} \text{L}^{f} \right\}$ $\quad 0 = -2 - 4e + f \quad f = 2 + 4e \quad f = -2$

The second Π is thus

$\Pi_2$:

$$\Pi_2 = \frac{P}{\rho c^2}$$

**Step 6** We write the final functional relationship as

*Relationship between* Π*s*:

$$\boxed{\frac{I}{\rho c^3} = f\left(\frac{P}{\rho c^2}\right)} \qquad (6)$$

***Discussion*** Equations 4 and 6 are the same. This exercise shows that you should get the same results using mass-based or force-based primary dimensions.

**7-109**

**Solution** We are to find the dimensionless relationship between the given parameters.

***Assumptions*** **1** The given parameters are the only relevant ones in the problem.

***Analysis*** The step-by-step method of repeating variables is employed to obtain the nondimensional parameters (the Πs).

**Step 1** There are now five parameters in this problem; $n = 5$,

*List of relevant parameters*: $$I = f\left(P, c, \rho, r\right) \qquad n = 5 \qquad (1)$$

**Step 2** The dimensions of $I$ are those of power per area. The primary dimensions of each parameter are listed,

$$\begin{array}{ccccc} I & P & c & \rho & r \\ \{m^1t^{-3}\} & \{m^1L^{-1}t^{-2}\} & \{L^1t^{-1}\} & \{m^1L^{-3}\} & \{L^1\} \end{array}$$

**Step 3** As a first guess, $j$ is set equal to 3, the number of primary dimensions represented in the problem (m, L, and t).

*Reduction*: $$j = 3$$

If this value of $j$ is correct, the expected number of Πs is

*Number of expected* Πs: $$k = n - j = 5 - 3 = 2$$

**Step 4** We need to choose three repeating parameters since $j = 3$. We pick the three simplest independent parameters ($r$ instead of $P$),

*Repeating parameters*: $r$, $\rho$, and $c$

**Step 5** The first Π is generated:

$$\Pi_1 = I \times r^a \rho^b c^c \qquad \{\Pi_1\} = \left\{\left(m^1t^{-3}\right)\left(L^1\right)^a\left(m^1L^{-3}\right)^b\left(L^1t^{-1}\right)^c\right\}$$

*mass:* $\{m^0\} = \{m^1m^b\}$ $\qquad 0 = 1 + b \qquad b = -1$

*time:* $\{t^0\} = \{t^{-3}t^{-c}\}$ $\qquad 0 = -3 - c \qquad c = -3$

*length:* $\{L^0\} = \{L^aL^{-3b}L^c\}$ $\qquad 0 = a - 3b + c$, $\quad a = 3b - c \qquad a = 0$

The first Π is thus

$\Pi_1$: $$\Pi_1 = \frac{I}{\rho c^3}$$

We form the second Π with sound pressure $P$,

$$\Pi_2 = P \times r^d \rho^e c^f \qquad \{\Pi_2\} = \left\{\left(m^1L^{-1}t^{-2}\right)\left(L^1\right)^d\left(m^1L^{-3}\right)^e\left(L^1t^{-1}\right)^f\right\}$$

*mass:* $\{m^0\} = \{m^1m^e\}$ $\qquad 0 = 1 + e \qquad e = -1$

*time:* $\{t^0\} = \{t^{-2}t^{-f}\}$ $\quad 0 = -2 - f \quad f = -2$

*length:* $\{L^0\} = \{L^{-1}L^{d}L^{-3e}L^{f}\}$ $\quad 0 = -1 + d - 3e + f$, $\; d = 1 + 3e - f \quad d = 0$

The second Π is thus

$\Pi_2$:
$$\Pi_2 = \frac{P}{\rho c^2}$$

**Step 6** We write the final functional relationship as

*Relationship between* Π*s*:
$$\boxed{\frac{I}{\rho c^3} = f\left(\frac{P}{\rho c^2}\right)} \qquad (2)$$

***Discussion*** This is an interesting case in which we added another independent parameter (*r*), yet this new parameter does not even appear in the final functional relationship! The list of independent parameters is thus *over specified*. (It turns out that *P* is a function of *r*, so *r* is not needed in the problem.) The result here is identical to the result of the previous problem. It turns out that the function in Eq. 2 is a constant times $\Pi_2^2$, which yields the correct analytical equation for *I*, namely

*Analytical result*:
$$I = \text{constant} \times \frac{P^2}{\rho c} \qquad (3)$$



# Solutions Manual for

# Fluid Mechanics: Fundamentals and Applications

# by Çengel & Cimbala

---

# CHAPTER 8
# FLOW IN PIPES

---

## Laminar and Turbulent Flow

### 8-1C

**Solution** We are to discuss why pipes are usually circular in cross section.

***Analysis*** Liquids are usually transported in circular pipes because pipes with a circular cross section **can withstand large pressure differences between the inside and the outside without undergoing any significant distortion**.

***Discussion*** Piping for gases at low pressure are often non-circular (e.g., air conditioning and heating ducts in buildings).

---

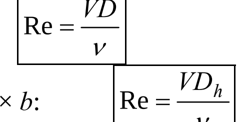
### 8-2C

**Solution** We are to define and discuss Reynolds number for pipe and duct flow.

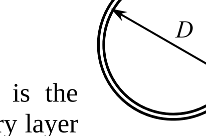
***Analysis*** *Reynolds number* is the **ratio of the inertial forces to viscous forces**, and it serves as a criterion for determining the flow regime. At *large* Reynolds numbers, for example, the flow is turbulent since the inertia forces are large relative to the viscous forces, and thus the viscous forces cannot prevent the random and rapid fluctuations of the fluid. It is defined as follows:

(*a*) For flow in a circular tube of inner diameter *D*: $\boxed{\mathrm{Re} = \frac{VD}{\nu}}$

(*b*) For flow in a rectangular duct of cross-section $a \times b$: $\boxed{\mathrm{Re} = \frac{VD_h}{\nu}}$



where $D_h = \frac{4A_c}{p} = \frac{4ab}{2(a+b)} = \frac{2ab}{(a+b)}$ is the hydraulic diameter.




***Discussion*** Since pipe flows become fully developed far enough downstream, diameter is the appropriate length scale for the Reynolds number. In boundary layer flows, however, the boundary layer grows continually downstream, and therefore downstream distance is a more appropriate length scale.

---

### 8-3C

**Solution** We are to compare the Reynolds number in air and water.

***Analysis*** Reynolds number is inversely proportional to kinematic viscosity, which is much smaller for water than for air (at 25°C, $\nu_{air}$ = 1.562×10$^{-5}$ m$^2$/s and $\nu_{water} = \mu/\rho$ = 0.891×10$^{-3}$/997 = 8.9×10$^{-7}$ m$^2$/s). Therefore, noting that Re = $VD/\nu$, the **Reynolds number is higher for motion in water for the same diameter and speed**.

***Discussion*** Of course, it is not possible to walk as fast in water as in air – try it!

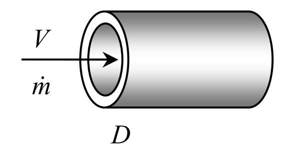
---

### 8-4C

**Solution** We are to express the Reynolds number for a circular pipe in terms of mass flow rate.

***Analysis*** Reynolds number for flow in a circular tube of diameter D is expressed as

$$\mathrm{Re} = \frac{VD}{\nu} \quad \text{where} \quad V = V_{avg} = \frac{\dot m}{\rho A_c} = \frac{\dot m}{\rho\left(\pi D^2/4\right)} = \frac{4\dot m}{\rho \pi D^2} \quad \text{and} \quad \nu = \frac{\mu}{\rho}$$




Substituting,

$$\mathrm{Re} = \frac{VD}{\nu} = \frac{4\dot m D}{\rho \pi D^2 (\mu/\rho)} = \frac{4\dot m}{\pi D \mu}. \text{ Thus, } \boxed{\mathrm{Re} = \frac{4\dot m}{\pi D \mu}}$$

***Discussion*** This result holds only for circular pipes.

---

**8-5C**

**Solution** We are to compare the pumping requirement for water and oil.

***Analysis*** **Engine oil requires a larger pump** because of its much larger viscosity.

***Discussion*** The density of oil is actually 10 to 15% smaller than that of water, and this makes the pumping requirement smaller for oil than water. However, the viscosity of oil is orders of magnitude larger than that of water, and is therefore the dominant factor in this comparison.

---

**8-6C**

**Solution** We are to discuss the Reynolds number for transition from laminar to turbulent flow.

***Analysis*** The generally accepted value of the Reynolds number above which the flow in a smooth pipe is turbulent is **4000**. **In the range 2300 < Re < 4000, the flow is typically transitional between laminar and turbulent**.

***Discussion*** In actual practice, pipe flow may become turbulent at Re lower or higher than this value.

---

**8-7C**

**Solution** We are to compare pipe flow in air and water.

***Analysis*** Reynolds number is inversely proportional to kinematic viscosity, which is much smaller for water than for air (at 25°C, $\nu_{air} = 1.562\times10^{-5}$ m$^2$/s and $\nu_{water} = \mu/\rho = 0.891\times10^{-3}/997 = 8.9\times10^{-7}$ m$^2$/s). Therefore, for the same diameter and speed, the Reynolds number will be higher for water flow, and thus **the flow is more likely to be turbulent for water**.

***Discussion*** The actual viscosity (dynamic viscosity) $\mu$ is larger for water than for air, but the density of water is so much greater than that of air that the kinematic viscosity of water ends up being smaller than that of air.

---

**8-8C**

**Solution** We are to define and discuss hydraulic diameter.

***Analysis*** For flow through non-circular tubes, the Reynolds number and the friction factor are based on the *hydraulic diameter* $D_h$ defined as $\boxed{D_h = \dfrac{4A_c}{p}}$ where $A_c$ is the cross-sectional area of the tube and $p$ is its perimeter. The hydraulic diameter is defined such that it **reduces to ordinary diameter *D* for circular tubes** since $\boxed{D_h = \dfrac{4A_c}{p} = \dfrac{4\pi D^2/4}{\pi D} = D}$.

***Discussion*** Hydraulic diameter is a useful tool for dealing with non-circular pipes (e.g., air conditioning and heating ducts in buildings).

---

**8-9C**

**Solution** We are to define and discuss hydrodynamic entry length.

***Analysis*** The **region from the tube inlet to the point at which the boundary layer merges at the centerline** is called the *hydrodynamic entrance region*, and the length of this region is called *hydrodynamic entry length*. **The entry length is much longer in laminar flow than it is in turbulent flow**. But at very low Reynolds numbers, $L_h$ is very small (e.g., $L_h = 1.2D$ at Re = 20).

***Discussion*** The entry length increases with increasing Reynolds number, but there is a significant change in entry length when the flow changes from laminar to turbulent.

---

**8-10C**

**Solution** We are to compare the wall shear stress at the inlet and outlet of a pipe.

***Analysis*** The wall shear stress $\tau_w$ is **highest at the tube inlet where the thickness of the boundary layer is nearly zero, and decreases gradually to the fully developed value**. The same is true for turbulent flow.

***Discussion*** We are assuming that the entrance is well-rounded so that the inlet flow is nearly uniform.

**8-11C**

**Solution** We are to discuss the effect of surface roughness on pressure drop in pipe flow.

***Analysis*** In turbulent flow, tubes with rough surfaces have much higher friction factors than the tubes with smooth surfaces, and thus **surface roughness leads to a much larger pressure drop in turbulent pipe flow**. In the case of laminar flow, the effect of surface roughness on the friction factor and pressure drop is negligible.

***Discussion*** The effect of roughness on pressure drop is significant for turbulent flow, as seen in the Moody chart.

## Fully Developed Flow in Pipes

**8-12C**

**Solution** We are to discuss how the wall shear stress varies along the flow direction in a pipe.

***Analysis*** The wall shear stress $\tau_w$ **remains constant along the flow direction in the fully developed region** in both laminar and turbulent flow.

***Discussion*** However, in the entrance region, $\tau_w$ starts out large, and decreases until the flow becomes fully developed.

**8-13C**

**Solution** We are to discuss the fluid property responsible for development of a velocity boundary layer.

***Analysis*** The fluid **viscosity** is responsible for the development of the velocity boundary layer.

***Discussion*** You can think of it this way: As the flow moves downstream, more and more of it gets slowed down near the wall due to friction, which is due to viscosity in the fluid.

**8-14C**

**Solution** We are to discuss the velocity profile in fully developed pipe flow.

***Analysis*** In the fully developed region of flow in a circular pipe, the velocity profile **does *not* change** in the flow direction.

***Discussion*** This is, in fact, the definition of fully developed – namely, the velocity profile remains of constant shape.

**8-15C**

**Solution** We are to discuss the relationship between friction factor and pressure loss in pipe flow.

***Analysis*** The f**riction factor for flow in a tube is proportional to the pressure loss**. Since the pressure loss along the flow is directly related to the power requirements of the pump to maintain flow, the **friction factor is also proportional to the power requirements to overcome friction**. The applicable relations are

$$\dot{W}_{\text{pump}} = \frac{\dot{m}\Delta P_L}{\rho} \quad \text{and} \quad \dot{W}_{\text{pump}} = \frac{\dot{m}\Delta P_L}{\rho}$$

***Discussion*** This type of pressure loss due to friction is an *irreversible* loss. Hence, it is always positive (positive being defined as a pressure drop down the pipe). A negative pressure loss would violate the second law of thermodynamics.

**8-16C**

**Solution** We are to discuss the value of shear stress at the center of a pipe.

***Analysis*** **The shear stress at the center of a circular tube during fully developed laminar flow is zero** since the shear stress is proportional to the velocity gradient, which is zero at the tube center.

***Discussion*** This result is due to the axisymmetry of the velocity profile.

**8-17C**

**Solution** We are to discuss whether the maximum shear stress in a turbulent pipe flow occurs at the wall.

***Analysis*** **Yes, the shear stress at the surface of a tube during fully developed turbulent flow is maximum** since the shear stress is proportional to the velocity gradient, which is maximum at the tube surface.

***Discussion*** This result is also true for laminar flow.

**8-18C**

**Solution** We are to discuss the change in head loss when the pipe length is doubled.

***Analysis*** In fully developed flow in a circular pipe with negligible entrance effects, if the length of the pipe is doubled, **the head loss also doubles** (the head loss is proportional to pipe length in the fully developed region of flow).

***Discussion*** If entrance lengths are not negligible, the head loss in the longer pipe would be less than twice that of the shorter pipe, since the shear stress is larger in the entrance region than in the fully developed region.

**8-19C**

**Solution** We are to examine a claim about volume flow rate in laminar pipe flow.

***Analysis*** **Yes**, the volume flow rate in a circular pipe with laminar flow can be determined by measuring the velocity at the centerline in the fully developed region, multiplying it by the cross-sectional area, and dividing the result by 2. This works for fully developed laminar pipe flow in round pipes since $\dot{V} = V_{\text{avg}} A_c = (V_{\max} / 2) A_c$.

***Discussion*** This is *not* true for turbulent flow, so one must be careful that the flow is laminar before trusting this measurement. It is also *not* true if the pipe is not round, even if the flow is fully developed and laminar.

**8-20C**

**Solution** We are to examine a claim about volume flow rate in laminar pipe flow.

***Analysis*** **No**, the average velocity in a circular pipe in fully developed laminar flow *cannot* be determined by simply measuring the velocity at $R/2$ (midway between the wall surface and the centerline). The average velocity is $V_{max}/2$, but the velocity at $R/2$ is

$$V(R/2) = V_{max}\left(1 - \frac{r^2}{R^2}\right)_{r=R/2} = \frac{3V_{max}}{4},$$ which is much larger than $V_{max}/2$.

***Discussion*** There is, of course, a radial location in the pipe at which the local velocity *is* equal to the average velocity. Can you find that location?

---

**8-21C**

**Solution** We are to compare the head loss when the pipe diameter is halved.

***Analysis*** In fully developed laminar flow in a circular pipe, the head loss is given by

$$h_L = f\frac{L}{D}\frac{V^2}{2g} = \frac{64}{\text{Re}}\frac{L}{D}\frac{V^2}{2g} = \frac{64}{V\,D/\nu}\frac{L}{D}\frac{V^2}{2g} = \frac{64\nu}{D}\frac{L}{D}\frac{V}{2g}$$

The average velocity can be expressed in terms of the flow rate as $V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4}$. Substituting,

$$h_L = \frac{64\nu}{D^2}\frac{L}{2g}\left(\frac{\dot{V}}{\pi D^2/4}\right) = \frac{64\nu}{D^2}\frac{4L\dot{V}}{2g\pi D^2} = \frac{128\nu L\dot{V}}{g\pi D^4}$$

Therefore, at constant flow rate and pipe length, the head loss is inversely proportional to the 4$^{th}$ power of diameter, and thus **reducing the pipe diameter by half increases the head loss by a factor of 16**.

***Discussion*** This is a very significant increase in head loss, and shows why larger diameter tubes lead to much smaller pumping power requirements.

---

**8-22C**

**Solution** We are to discuss why the friction factor is higher in turbulent pipe flow compared to laminar pipe flow.

***Analysis*** In turbulent flow, it is the **turbulent eddies due to enhanced mixing** that cause the friction factor to be larger. This turbulent mixing leads to a much larger wall shear stress, which translates into larger friction factor.

***Discussion*** Another way to think of it is that the turbulent eddies cause the turbulent velocity profile to be much *fuller* (closer to uniform flow) than the laminar velocity profile.

---

**8-23C**

**Solution** We are to define and discuss turbulent viscosity.

***Analysis*** Turbulent viscosity $\mu_t$ is caused by turbulent eddies, and it **accounts for momentum transport by turbulent eddies**. It is expressed as $\boxed{\tau_t = -\rho\overline{u'v'} = \mu_t\frac{\partial\bar{u}}{\partial y}}$ where $\bar{u}$ is the mean value of velocity in the flow direction and $u'$ and $u'$ are the fluctuating components of velocity.

***Discussion*** Turbulent viscosity is a derived, or non-physical quantity. Unlike the viscosity, it is *not* a property of the fluid; rather, it is a property of the *flow*.

---

**8-24C**

**Solution** We are to discuss the dimensions of a constant in a head loss expression.

***Analysis*** We compare the dimensions of the two sides of the equation $h_L = 0.0826\, fL\frac{\dot{V}^2}{D^5}$. Using curly brackets to mean "the dimensions of", we have $\{L\} = \{0.0826\}\cdot\{1\}\{L\}\cdot\{L^3t^{-1}\}^2\cdot\{L^{-5}\}$, and the dimensions of the constant are thus $\{0.0826\} = \{L^{-1}t^2\}$. Therefore, **the constant 0.0826 is *not* dimensionless**. **This is *not* a dimensionally homogeneous equation**, and it cannot be used in any consistent set of units.

***Discussion*** Engineers often create dimensionally inhomogeneous equations like this. While they are useful for practicing engineers, they are valid only when the proper units are used for each variable, and this can occasionally lead to mistakes. For this reason, the present authors do not encourage their use.

---

**8-25C**

**Solution** We are to discuss the change in head loss due to a decrease in viscosity by a factor of two.

***Analysis*** In fully developed laminar flow in a circular pipe, the pressure loss and the head loss are given by

$$\Delta P_L = \frac{32\mu LV}{D^2} \quad \text{and} \quad h_L = \frac{\Delta P_L}{\rho g} = \frac{32\mu LV}{\rho g D^2}$$

When the flow rate and thus the average velocity are held constant, the head loss becomes proportional to viscosity. Therefore, **the head loss is reduced by half** when the viscosity of the fluid is reduced by half.

***Discussion*** This result is *not* valid for turbulent flow – only for laminar flow. It is also not valid for laminar flow in situations where the entrance length effects are not negligible.

---

**8-26C**

**Solution** We are to discuss the relationship between head loss and pressure drop in pipe flow.

***Analysis*** The head loss is related to pressure loss by $\boxed{h_L = \Delta P_L / \rho g}$. For a given fluid, the head loss can be converted to pressure loss by multiplying the head loss by the acceleration of gravity and the density of the fluid. Thus, for constant density, **head loss and pressure drop are linearly proportional to each other**.

***Discussion*** This result is true for both laminar and turbulent pipe flow.

---

**8-27C**

**Solution** We are to discuss if the friction factor is zero for laminar pipe flow with a perfectly smooth surface.

***Analysis*** During laminar flow of air in a circular pipe with perfectly smooth surfaces, **the friction factor is *not* zero** because of the no-slip boundary condition, which must hold even for perfectly smooth surfaces.

***Discussion*** If we compare the friction factor for rough and smooth surfaces, roughness has no effect on friction factor for fully developed laminar pipe flow unless the roughness height is very large. For turbulent pipe flow, however, roughness very strongly impacts the friction factor.

---

**8-28C**

**Solution** We are to explain why friction factor is independent of Re at very large Re.

***Analysis*** At very large Reynolds numbers, the flow is fully rough and the friction factor is independent of the Reynolds number. This is because **the thickness of viscous sublayer decreases with increasing Reynolds number, and it be comes so thin that the surface roughness protrudes into the flow**. The viscous effects in this case are produced in the main flow primarily by the protruding roughness elements, and the contribution of the viscous sublayer is negligible.

***Discussion*** This effect is clearly seen in the Moody chart – at large Re, the curves flatten out horizontally.

---

**8-29E**

**Solution** The pressure readings across a pipe are given. The flow rates are to be determined for three different orientations of horizontal, uphill, and downhill flow.

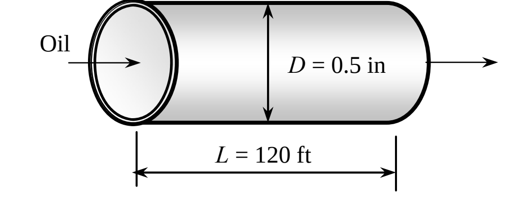
***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The flow is laminar (to be verified). **4** The pipe involves no components such as bends, valves, and connectors. **5** The piping section involves no work devices such as pumps and turbines.

***Properties*** The density and dynamic viscosity of oil are given to be $\rho$ = 56.8 lbm/ft$^3$ and $\mu$ = 0.0278 lbm/ft·s, respectively.

***Analysis*** The pressure drop across the pipe and the cross-sectional area of the pipe are

$$\Delta P = P_1 - P_2 = 120 - 14 = 106 \text{ psi}$$

$$A_c = \pi D^2/4 = \pi(0.5/12 \text{ ft})^2/4 = 0.001364 \text{ ft}^2$$




(*a*) The flow rate for all three cases can be determined from

$$\dot{V} = \frac{(\Delta P - \rho g L \sin\theta)\pi D^4}{128\mu L}$$

where θ is the angle the pipe makes with the horizontal. For the horizontal case, θ = 0 and thus sin θ = 0. Therefore,

$$\dot{V}_{\text{horiz}} = \frac{\Delta P \pi D^4}{128\mu L} = \frac{(106 \text{ psi})\pi(0.5/12 \text{ ft})^4}{128(0.0278 \text{ lbm/ft}\cdot\text{s})(120 \text{ ft})}\left(\frac{144 \text{ lbf/ft}^2}{1 \text{ psi}}\right)\left(\frac{32.2 \text{ lbm}\cdot\text{ft/s}^2}{1 \text{ lbf}}\right) = \mathbf{0.0109 \text{ ft}^3/\text{s}}$$

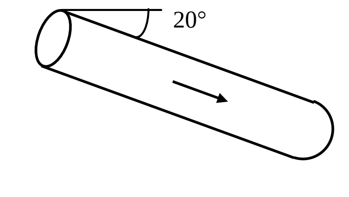
(*b*) For uphill flow with an inclination of 20°, we have θ = +20°, and

$$\rho g L \sin\theta = (56.8 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)(120 \text{ ft})\sin 20°\left(\frac{1 \text{ psi}}{144 \text{ lbf/ft}^2}\right)\left(\frac{1 \text{ lbf}}{32.2 \text{ lbm}\cdot\text{ft/s}^2}\right) = 16.2 \text{ psi}$$

$$\dot{V}_{\text{uphill}} = \frac{(\Delta P - \rho g L \sin\theta)\pi D^4}{128\mu L} =$$

$$= \frac{(106 - 16.2 \text{ psi})\pi(0.5/12 \text{ ft})^4}{128(0.0278 \text{ lbm/ft}\cdot\text{s})(120 \text{ ft})}\left(\frac{144 \text{ lbf/ft}^2}{1 \text{ psi}}\right)\left(\frac{32.2 \text{ lbm}\cdot\text{ft/s}^2}{1 \text{ lbf}}\right) = \mathbf{0.00923 \text{ ft}^3/\text{s}}$$

(*c*) For downhill flow with an inclination of 20°, we have θ = -20°, and




$$\dot{V}_{\text{downhill}} = \frac{(\Delta P - \rho g L \sin\theta)\pi D^4}{128\mu L}$$

$$= \frac{[106 - (-16.2) \text{ psi}]\pi(0.5/12 \text{ ft})^4}{128(0.0278 \text{ lbm/ft}\cdot\text{s})(120 \text{ ft})}\left(\frac{144 \text{ lbf/ft}^2}{1 \text{ psi}}\right)\left(\frac{32.2 \text{ lbm}\cdot\text{ft/s}^2}{1 \text{ lbf}}\right) = \mathbf{0.0126 \text{ ft}^3/\text{s}}$$

The flow rate is the highest for downhill flow case, as expected. The average fluid velocity and the Reynolds number in this case are

$$V = \frac{\dot{V}}{A_c} = \frac{0.0126 \text{ ft}^3/\text{s}}{0.001364 \text{ ft}^2} = 9.24 \text{ ft/s}$$

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(56.8 \text{ lbm/ft}^3)(9.24 \text{ ft/s})(0.5/12 \text{ ft})}{0.0278 \text{ lbm/ft}\cdot\text{s}} = 787$$

which is less than 2300. Therefore, the flow is **laminar** for all three cases, and the analysis above is valid.

***Discussion*** Note that the flow is driven by the combined effect of pressure difference and gravity. As can be seen from the calculated rates above, gravity opposes uphill flow, but helps downhill flow. Gravity has no effect on the flow rate in the horizontal case. Downhill flow can occur even in the absence of an applied pressure difference.

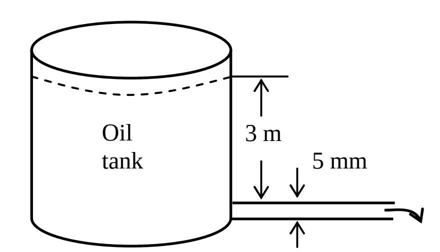
**8-30**

**Solution** Oil is being discharged by a horizontal pipe from a storage tank open to the atmosphere. The flow rate of oil through the pipe is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The entrance and exit loses are negligible. **4** The flow is laminar (to be verified). **5** The pipe involves no components such as bends, valves, and connectors. **6** The piping section involves no work devices such as pumps and turbines.

***Properties*** The density and kinematic viscosity of oil are given to be $\rho$ = 850 kg/m$^3$ and $\nu$ = 0.00062 m$^2$/s, respectively. The dynamic viscosity is calculated to be

$$\mu = \rho\nu = (850\ \text{kg/m}^3)(0.00062\ \text{m}^2/s) = 0.527\ \text{kg/m}\cdot\text{s}$$




***Analysis*** The pressure at the bottom of the tank is

$$P_{1,\text{gage}} = \rho g h$$

$$= (850\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(3\ \text{m})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)$$

$$= 25.02\ \text{kN/m}^2$$

Disregarding inlet and outlet losses, the pressure drop across the pipe is

$$\Delta P = P_1 - P_2 = P_1 - P_{\text{atm}} = P_{1,\text{gage}} = 25.02\ \text{kN/m}^2 = 25.02\ \text{kPa}$$

The flow rate through a horizontal pipe in laminar flow is determined from

$$\dot{V}_{\text{horiz}} = \frac{\Delta P \pi D^4}{128\mu L} = \frac{(25.02\ \text{kN/m}^2)\pi(0.005\ \text{m})^4}{128(0.527\ \text{kg/m}\cdot\text{s})(40\ \text{m})}\left(\frac{1000\ \text{kg}\cdot\text{m/s}^2}{1\ \text{kN}}\right) = 1.821\times10^{-8}\ \text{m}^3/\text{s} \cong \mathbf{1.82\times10^{-8}\ m^3/s}$$

The average fluid velocity and the Reynolds number in this case are

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{1.821\times10^{-8}\ \text{m}^3/\text{s}}{\pi(0.005\ \text{m})^2/4} = 9.27\times10^{-4}\ \text{m/s}$$

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(850\ \text{kg/m}^3)(9.27\times10^{-4}\ \text{m/s})(0.005\ \text{m})}{0.527\ \text{kg/m}\cdot\text{s}} = 0.0075$$

which is less than 2300. Therefore, the flow is *laminar* and the analysis above is valid.

***Discussion*** The flow rate will be even less when the inlet and outlet losses are considered, especially when the inlet is not well-rounded.

**8-31**

**Solution** The average flow velocity in a pipe is given. The pressure drop, the head loss, and the pumping power are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The pipe involves no components such as bends, valves, and connectors. **4** The piping section involves no work devices such as pumps and turbines.

***Properties*** The density and dynamic viscosity of water are given to be $\rho$ = 999.7 kg/m$^3$ and $\mu$ = 1.307×10$^{-3}$ kg/m·s, respectively.

***Analysis*** (*a*) First we need to determine the flow regime. The Reynolds number of the flow is

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(999.7\ \text{kg/m}^3)(1.2\ \text{m/s})(2\times10^{-3}\ \text{m})}{1.307\times10^{-3}\ \text{kg/m}\cdot\text{s}} = 1836$$



which is less than 2300. Therefore, the flow is laminar. Then the friction factor and the pressure drop become

$$f = \frac{64}{\text{Re}} = \frac{64}{1836} = 0.0349$$

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.0349\frac{15\ \text{m}}{0.002\ \text{m}}\frac{(999.7\ \text{kg/m}^3)(1.2\ \text{m/s})^2}{2}\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}}\right)\left(\frac{1\ \text{kPa}}{1\ \text{kN/m}^2}\right) = \mathbf{188\ kPa}$$

(*b*) The head loss in the pipe is determined from

$$h_L = \frac{\Delta P_L}{\rho g} = f\frac{L}{D}\frac{V^2}{2g} = 0.0349\frac{15\ \text{m}}{0.002\ \text{m}}\frac{(1.2\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)} = \mathbf{19.2\ m}$$

(*c*) The volume flow rate and the pumping power requirements are

$$\dot{V} = VA_c = V(\pi D^2/4) = (1.2\ \text{m/s})[\pi(0.002\ \text{m})^2/4] = 3.77\times10^{-6}\ \text{m}^3/s$$

$$\dot{W}_{\text{pump}} = \dot{V}\Delta P = (3.77\times10^{-6}\ \text{m}^3/s)(188\ \text{kPa})\left(\frac{1000\ \text{W}}{1\ \text{kPa}\cdot\text{m}^3/\text{s}}\right) = \mathbf{0.71\ W}$$

Therefore, power input in the amount of 0.71 W is needed to overcome the frictional losses in the flow due to viscosity.

***Discussion*** If the flow were instead *turbulent*, the pumping power would be much greater since the head loss in the pipe would be much greater.

**8-32**

**Solution** The flow rate through a specified water pipe is given. The pressure drop, the head loss, and the pumping power requirements are to be determined.

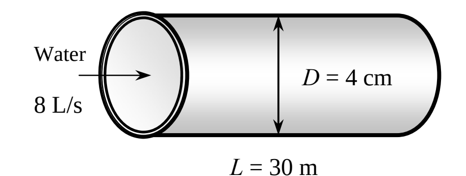
***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The pipe involves no components such as bends, valves, and connectors. **4** The piping section involves no work devices such as pumps and turbines.

***Properties*** The density and dynamic viscosity of water are given to be $\rho$ = 999.1 kg/m$^3$ and $\mu$ = 1.138×10$^{-3}$ kg/m·s, respectively. The roughness of stainless steel is 0.002 mm.

***Analysis*** First we calculate the average velocity and the Reynolds number to determine the flow regime:

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.008\ \text{m}^3/s}{\pi(0.04\ \text{m})^2/4} = 6.366\ \text{m}/s$$

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(999.1\ \text{kg/m}^3)(6.366\ \text{m/s})(0.04\ \text{m})}{1.138\times10^{-3}\ \text{kg/m}\cdot\text{s}} = 2.236\times10^5$$




which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D = \frac{2\times10^{-6}\ \text{m}}{0.04\ \text{m}} = 5\times10^{-5}$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{5\times10^{-5}}{3.7} + \frac{2.51}{2.236\times10^5\sqrt{f}}\right)$$

It gives $f$ = 0.01573. Then the pressure drop, head loss, and the required power input become

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.01573\frac{30\ \text{m}}{0.04\ \text{m}}\frac{(999.1\ \text{kg/m}^3)(6.366\ \text{m/s})^2}{2}\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}}\right)\left(\frac{1\ \text{kPa}}{1\ \text{kN/m}^2}\right) = \mathbf{239\ kPa}$$

$$h_L = \frac{\Delta P_L}{\rho g} = f\frac{L}{D}\frac{V^2}{2g} = 0.01573\frac{30\ \text{m}}{0.04\ \text{m}}\frac{(6.366\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)} = \mathbf{24.4\ m}$$

$$\dot{W}_{\text{pump}} = \dot{V}\Delta P = (0.008\ \text{m}^3/s)(239\ \text{kPa})\left(\frac{1\ \text{kW}}{1\ \text{kPa}\cdot\text{m}^3/\text{s}}\right) = \mathbf{1.91\ kW}$$

Therefore, useful power input in the amount of 1.91 kW is needed to overcome the frictional losses in the pipe.

***Discussion*** The friction factor could also be determined easily from the explicit Haaland relation. It would give $f$ = 0.0155, which is sufficiently close to 0.0157. Also, the friction factor corresponding to $\varepsilon$ = 0 in this case is 0.0153, which indicates that stainless steel pipes in this case can be assumed to be smooth with an error of about 2%. Also, the power input determined is the mechanical power that needs to be imparted to the fluid. The shaft power will be more than this due to pump inefficiency; the electrical power input will be even more due to motor inefficiency.

**8-33E**

**Solution** The flow rate and the head loss in an air duct is given. The minimum diameter of the duct is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The duct involves no components such as bends, valves, and connectors. **4** Air is an ideal gas. **5** The duct is smooth since it is made of plastic, $\varepsilon \approx 0$. **6** The flow is turbulent (to be verified).

***Properties*** The density, dynamic viscosity, and kinematic viscosity of air at 100°F are $\rho$ = 0.07088 lbm/ft$^3$, $\mu$ = 0.04615 lbm/ft·h, and $\nu$ = 0.6512 ft$^2$/s = 1.809×10$^{-4}$ ft$^2$/s.

***Analysis*** The average velocity, Reynolds number, friction factor, and the head loss relations can be expressed as ($D$ is in ft, $V$ is in ft/s, Re and $f$ are dimensionless)

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{12 \text{ ft}^3/s}{\pi D^2/4}$$

$$\text{Re} = \frac{VD}{\nu} = \frac{VD}{1.809\times10^{-4} \text{ ft}^2/s}$$

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) = -2.0\log\left(\frac{2.51}{\text{Re}\sqrt{f}}\right)$$

$$h_L = f\frac{L}{D}\frac{V^2}{2g} \quad \rightarrow \quad 50 = f\frac{L}{D}\frac{V^2}{2g} = f\frac{400 \text{ ft}}{D}\frac{V^2}{2(32.2 \text{ ft/s}^2)}$$

Air

12 ft$^3$/s

$D$

$L$ = 400 ft

This is a set of 4 equations in 4 unknowns, and solving them with an equation solver gives

$D$ = **0.88 ft**, $f$ = 0.0181, $V$ = 19.8 ft/s, and Re = 96,040

Therefore, the diameter of the duct should be more than 0.88 ft if the head loss is not to exceed 50 ft. Note that Re > 4000, and thus the turbulent flow assumption is verified.

The diameter can also be determined directly from the third Swamee-Jain formula to be

$$D = 0.66\left[\varepsilon^{1.25}\left(\frac{L\dot{V}^2}{gh_L}\right)^{4.75} + \nu\dot{V}^{9.4}\left(\frac{L}{gh_L}\right)^{5.2}\right]^{0.04}$$

$$= 0.66\left[0 + (0.180\times10^{-3} \text{ ft}^2/s)(12 \text{ ft}^3/s)^{9.4}\left(\frac{400 \text{ ft}}{(32.2 \text{ ft/s}^2)(50 \text{ ft})}\right)^{5.2}\right]^{0.04}$$

$$= 0.89 \text{ ft}$$

***Discussion*** Note that the difference between the two results is less than 2%. Therefore, the simple Swamee-Jain relation can be used with confidence.

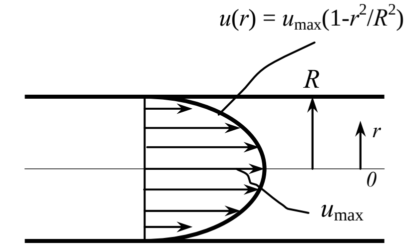
**8-34**

**Solution** In fully developed laminar flow in a circular pipe, the velocity at $r = R/2$ is measured. The velocity at the center of the pipe ($r = 0$) is to be determined.

***Assumptions*** The flow is steady, laminar, and fully developed.

***Analysis*** The velocity profile in fully developed laminar flow in a circular pipe is given by

$$u(r) = u_{\max}\left(1 - \frac{r^2}{R^2}\right)$$




where $u_{max}$ is the maximum velocity which occurs at pipe center, $r = 0$. At $r = R/2$,

$$u(R/2) = u_{\max}\left(1 - \frac{(R/2)^2}{R^2}\right) = u_{\max}\left(1 - \frac{1}{4}\right) = \frac{3u_{\max}}{4}$$

Solving for $u_{max}$ and substituting,

$$u_{max} = \frac{4u(R/2)}{3} = \frac{4(6 \text{ m/s})}{3} = \mathbf{8.00 \text{ m/s}}$$

which is the velocity at the pipe center.

***Discussion*** The relationship used here is valid only for fully developed laminar flow. The result would be much different if the flow were turbulent.

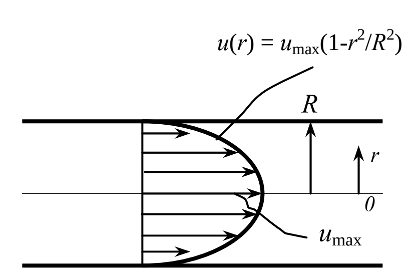
**8-35**

**Solution** The velocity profile in fully developed laminar flow in a circular pipe is given. The average and maximum velocities as well as the flow rate are to be determined.

***Assumptions*** The flow is steady, laminar, and fully developed.

***Analysis*** The velocity profile in fully developed laminar flow in a circular pipe is given by

$$u(r) = u_{\max}\left(1 - \frac{r^2}{R^2}\right)$$




The velocity profile in this case is given by

$$u(r) = 4(1 - r^2/R^2)$$

Comparing the two relations above gives the maximum velocity to be $\mathbf{u_{max} = 4.00 \text{ m/s}}$. Then the average velocity and volume flow rate become

$$V_{avg} = \frac{u_{max}}{2} = \frac{4 \text{ m/s}}{2} = \mathbf{2.00 \text{ m/s}}$$

$$\dot{V} = V_{avg} A_c = V_{avg}(\pi R^2) = (2 \text{ m/s})[\pi(0.02 \text{ m})^2] = \mathbf{0.00251 \text{ m}^3/\text{s}}$$

***Discussion*** A unique feature of fully developed laminar pipe flow is that the maximum velocity is exactly twice the average velocity. This is *not* the case for turbulent pipe flow, since the velocity profile is much fuller.

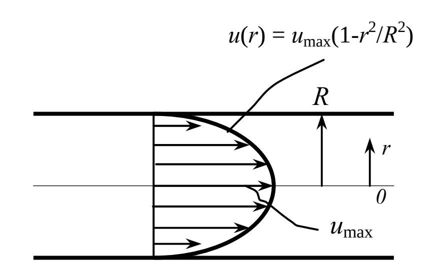
**8-36**

**Solution** The velocity profile in fully developed laminar flow in a circular pipe is given. The average and maximum velocities as well as the flow rate are to be determined.

***Assumptions*** The flow is steady, laminar, and fully developed.

***Analysis*** The velocity profile in fully developed laminar flow in a circular pipe is given by

$$u(r) = u_{\max}\left(1 - \frac{r^2}{R^2}\right)$$




The velocity profile in this case is given by

$$u(r) = 4(1 - r^2 / R^2)$$

Comparing the two relations above gives the maximum velocity to be $u_{max}$ **= 4.00 m/s**. Then the average velocity and volume flow rate become

$$V_{avg} = \frac{u_{max}}{2} = \frac{4 \text{ m/s}}{2} = \mathbf{2.00 \ m/s}$$

$$\dot{V} = V_{avg} A_c = V_{avg}(\pi R^2) = (2 \text{ m/s})[\pi (0.07 \text{ m})^2] = \mathbf{0.0308 \ m^3/s}$$

***Discussion*** Compared to the previous problem, the average velocity remains the same since the maximum velocity (at the centerline) remains the same, but the volume flow rate increases as the diameter increases.

---

**8-37**

**Solution** Air enters the constant spacing between the glass cover and the plate of a solar collector. The pressure drop of air in the collector is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The roughness effects are negligible, and thus the inner surfaces are considered to be smooth, $\varepsilon \approx 0$. **4** Air is an ideal gas. **5** The local atmospheric pressure is 1 atm.

***Properties*** The properties of air at 1 atm and 45° are $\rho = 1.109$ kg/m$^3$, $\mu = 1.941\times10^{-5}$ kg/m·s, and $\nu = 1.750\times10^{-5}$ m$^2$/s.

***Analysis*** Mass flow rate, cross-sectional area, hydraulic diameter, average velocity, and the Reynolds number are

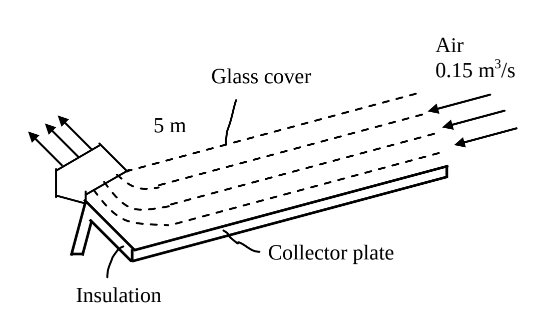
$$\dot{m} = \rho\dot{V} = (1.11 \text{ kg/m}^3)(0.15 \text{ m}^3/\text{s}) = 0.1665 \text{ kg/s}$$

$$A_c = a \times b = (1 \text{ m})(0.03 \text{ m}) = 0.03 \text{ m}^2$$

$$D_h = \frac{4A_c}{p} = \frac{4(0.03 \text{ m}^2)}{2(1+0.03) \text{ m}} = 0.05825 \text{ m}$$

$$V = \frac{\dot{V}}{A_c} = \frac{0.15 \text{ m}^3/s}{0.03 \text{ m}^2} = 5 \text{ m}/s$$

$$\text{Re} = \frac{VD_h}{\upsilon} = \frac{(5 \text{ m/s})(0.05825 \text{ m})}{1.750\times10^{-5} \text{ m}^2/\text{s}} = 1.664\times10^4$$




Since Re is greater than 4000, the flow is turbulent. The friction factor corresponding to this Reynolds number for a smooth flow section ($\varepsilon/D = 0$) can be obtained from the Moody chart. But to avoid reading error, we use the Colebrook equation,

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(0 + \frac{2.51}{16{,}640\sqrt{f}}\right)$$

which gives $f = 0.0271$. Then the pressure drop becomes

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.0271\frac{5 \text{ m}}{0.05825 \text{ m}}\frac{(1.11 \text{ kg/m}^3)(5 \text{ m/s})^2}{2}\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1 \text{ Pa}}{1 \text{ N/m}^2}\right) = \mathbf{32.3 \ Pa}$$

***Discussion*** The friction factor could also be determined easily from the explicit Haaland relation. It would give $f$ = 0.0270, which is sufficiently close to 0.0271.

---

**8-38**

**Solution** Oil flows through a pipeline that passes through icy waters of a lake. The pumping power needed to overcome pressure losses is to be determined.

***Assumptions*** The flow is steady and incompressible. **2** The flow section considered is away from the entrance, and thus the flow is fully developed. **3** The roughness effects are negligible, and thus the inner surfaces are considered to be smooth, $\varepsilon \approx 0$.

***Properties*** The properties of oil are given to be $\rho$ = 894 kg/m$^3$ and $\mu$ = 2.33 kg/m·s.

***Analysis*** The volume flow rate and the Reynolds number in this case are

(Icy lake, 0°C)

Oil
10°C
0.5 m/s

$D$ = 0.4 m

$L$ = 300 m

$$\dot{V} = VA_c = V\frac{\pi D^2}{4} = (0.5\text{ m/s})\frac{\pi(0.4\text{ m})^2}{4} = 0.0628\text{ m}^3/\text{s}$$

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(894\text{ kg/m}^3)(0.5\text{ m/s})(0.4\text{ m})}{2.33\text{ kg/m}\cdot\text{s}} = 76.7$$

which is less than 2300. Therefore, the flow is laminar, and the friction factor is

$$f = \frac{64}{\text{Re}} = \frac{64}{76.7} = 0.834$$

Then the pressure drop in the pipe and the required pumping power become

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.834\frac{300\text{ m}}{0.4\text{ m}}\frac{(894\text{ kg/m}^3)(0.5\text{ m/s})^2}{2}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right) = 69.9\text{ kPa}$$

$$\dot{W}_{\text{pump}} = \dot{V}\Delta P = (0.0628\text{ m}^3/\text{s})(69.9\text{ kPa})\left(\frac{1\text{ kW}}{1\text{ kPa}\cdot\text{m}^3/\text{s}}\right) = \mathbf{4.39\text{ kW}}$$

***Discussion*** The power input determined is the mechanical power that needs to be imparted to the fluid. The shaft power will be much more than this due to pump inefficiency; the electrical power input will be even more due to motor inefficiency.

---

**8-39**

**Solution** Laminar flow through a square channel is considered. The change in the head loss is to be determined when the average velocity is doubled.

***Assumptions*** **1** The flow remains laminar at all times. **2** The entrance effects are negligible, and thus the flow is fully developed.

***Analysis*** The friction factor for fully developed laminar flow in a square channel is

V

$$f = \frac{56.92}{\text{Re}} \quad \text{where} \quad \text{Re} = \frac{\rho VD}{\mu}$$

Then the head loss for laminar flow can be expressed as

$$h_{L,1} = f\frac{L}{D}\frac{V^2}{2g} = \frac{56.92}{\text{Re}}\frac{L}{D}\frac{V^2}{2g} = \frac{56.92\mu}{\rho VD}\frac{L}{D}\frac{V^2}{2g} = 28.46V\frac{\mu L}{\rho g D^2}$$

which shows that the head loss is proportional to the average velocity. Therefore, **the head loss doubles when the average velocity is doubled**. This can also be shown as

$$h_{L,2} = 28.46V_2\frac{\mu L}{\rho g D^2} = 28.46(2V)\frac{\mu L}{\rho g D^2} = 2\left(28.46V\frac{\mu L}{\rho g D^2}\right) = 2h_{L,1}$$

***Discussion*** The conclusion above is also valid for laminar flow in channels of different cross-sections.

---

**8-40**

**Solution** Turbulent flow through a smooth pipe is considered. The change in the head loss is to be determined when the average velocity is doubled.

***Assumptions*** **1** The flow remains turbulent at all times. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The inner surface of the pipe is smooth.

***Analysis*** The friction factor for the turbulent flow in smooth pipes is given as

$$f = 0.184\,\mathrm{Re}^{-0.2} \quad \text{where} \quad \mathrm{Re} = \frac{\rho VD}{\mu}$$

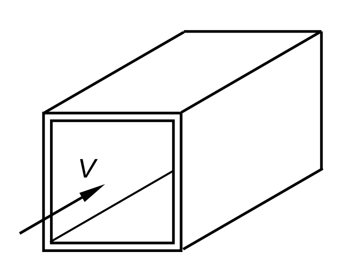
Then the head loss of the fluid for turbulent flow can be expressed as

$$h_{L,1} = f\frac{L}{D}\frac{V^2}{2g} = 0.184\,\mathrm{Re}^{-0.2}\frac{L}{D}\frac{V^2}{2g} = 0.184\left(\frac{\rho VD}{\mu}\right)^{-0.2}\frac{L}{D}\frac{V^2}{2g} = 0.184\left(\frac{\rho D}{\mu}\right)^{-0.2}\frac{L}{D}\frac{V^{1.8}}{2g}$$

which shows that the head loss is proportional to the 1.8$^{\text{th}}$ power of the average velocity. Therefore, the head loss increases by a factor of $2^{1.8} = 3.48$ when the average velocity is doubled. This can also be shown as

$$h_{L,2} = 0.184\left(\frac{\rho D}{\mu}\right)^{-0.2}\frac{L}{D}\frac{V_2^{1.8}}{2g} = 0.184\left(\frac{\rho D}{\mu}\right)^{-0.2}\frac{L}{D}\frac{(2V)^{1.8}}{2g}$$

$$= 2^{1.8}\left[0.184\left(\frac{\rho D}{\mu}\right)^{-0.2}\frac{L}{D}\frac{V^{1.8}}{2g}\right] = 2^{1.8}h_{L,1} = 3.48h_{L,1}$$




For fully rough flow in a rough pipe, the friction factor is independent of the Reynolds number and thus the flow velocity. Therefore, **the head loss increases by a factor of 4** in this case since

$$h_{L,1} = f\frac{L}{D}\frac{V^2}{2g}$$

and thus the **head loss is proportional to the square of the average velocity** when *f, L,* and *D* are constant.

***Discussion*** Most flows in practice are in the fully rough regime, and thus the head loss is generally assumed to be proportional to the square of the average velocity for all kinds of turbulent flow. Note that we use diameter *D* here in place of hydraulic diameter $D_h$. For a square duct, it turns out that $D_h = D$, so this is a valid approximation.

**8-41**

**Solution** Air enters a rectangular duct. The fan power needed to overcome the pressure losses is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** Air is an ideal gas. **4** The duct involves no components such as bends, valves, and connectors. **5** The flow section involves no work devices such as fans or turbines

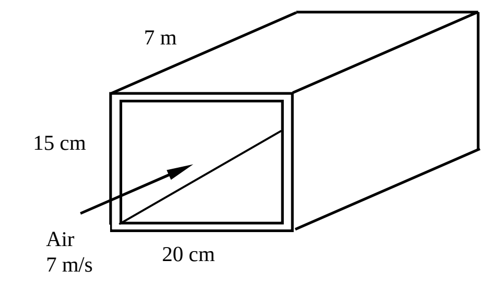
***Properties*** The properties of air at 1 atm and 35°C are $\rho$ = 1.145 kg/m$^3$, $\mu$ = 1.895×10$^{-5}$ kg/m⋅s, and $\nu$ = 1.655×10$^{-5}$ m$^2$/s. The roughness of commercial steel surfaces is $\varepsilon$ = 0.000045 m.

***Analysis*** The hydraulic diameter, the volume flow rate, and the Reynolds number in this case are

$$D_h = \frac{4A_c}{p} = \frac{4ab}{2(a+b)} = \frac{4(0.15\text{ m})(0.20\text{ m})}{2(0.15+0.20)\text{ m}} = 0.1714\text{ m}$$

$$\dot{V} = VA_c = V(a\times b) = (7\text{ m/s})(0.15\times 0.20\text{ m}^2) = 0.21\text{ m}^3\text{/s}$$

$$\text{Re} = \frac{\rho VD_h}{\mu} = \frac{(1.145\text{ kg/m}^3)(7\text{ m/s})(0.1714\text{ m})}{1.895\times 10^{-5}\text{ kg/m}\cdot\text{s}} = 72{,}490$$




which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon / D_h = \frac{4.5\times 10^{-5}\text{ m}}{0.1714\text{ m}} = 2.625\times 10^{-4}$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon / D_h}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{2.625\times 10^{-4}}{3.7} + \frac{2.51}{72{,}490\sqrt{f}}\right)$$

It gives $f$ = 0.02034. Then the pressure drop in the duct and the required pumping power become

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.02034\frac{7\text{ m}}{0.1714\text{ m}}\frac{(1.145\text{ kg/m}^3)(7\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ Pa}}{1\text{ N/m}^2}\right) = 23.3\text{ Pa}$$

$$\dot{W}_{\text{pump}} = \dot{V}\Delta P = \left(0.21\text{ m}^3\text{/s}\right)\left(23.3\text{ Pa}\right)\left(\frac{1\text{ W}}{1\text{ Pa}\cdot\text{m}^3\text{/s}}\right) = \mathbf{4.90\ W}$$

***Discussion*** The friction factor could also be determined easily from the explicit Haaland relation. It would give $f$ = 0.02005, which is sufficiently close to 0.02034. Also, the power input determined is the mechanical power that needs to be imparted to the fluid. The shaft power will be much more than this due to fan inefficiency; the electrical power input will be even more due to motor inefficiency.

**8-42E**

**Solution** Water passes through copper tubes at a specified rate. The pumping power required per ft length to maintain flow is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The pipe involves no components such as bends, valves, and connectors. **4** The piping section involves no work devices such as pumps and turbines.

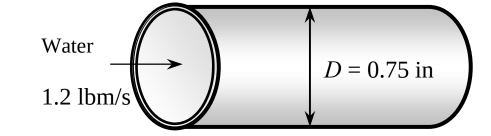
***Properties*** The density and dynamic viscosity of water at 60°F are $\rho$ = 62.36 lbm/ft$^3$ and $\mu$ = 2.713 lbm/ft·h = 7.536×10$^{-4}$ lbm/ft·s. The roughness of copper tubing is 5×10$^{-6}$ ft.

***Analysis*** First we calculate the average velocity and the Reynolds number to determine the flow regime:

$$V = \frac{\dot{m}}{\rho A_c} = \frac{\dot{m}}{\rho(\pi D^2/4)} = \frac{1.2\ \text{lbm}/s}{(62.36\ \text{lbm/ft}^3)[\pi(0.75/12\ \text{ft})^2/4]} = 6.272\ \text{ft/s}$$

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(62.36\ \text{lbm/ft}^3)(6.272\ \text{ft/s})(0.75/12\ \text{ft})}{7.536\times10^{-4}\ \text{lbm/ft}\cdot\text{s}} = 32{,}440$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is




$$\varepsilon/D = \frac{5\times10^{-6}\ \text{ft}}{0.75/12\ \text{ft}} = 8\times10^{-5}$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{8\times10^{-5}}{3.7} + \frac{2.51}{32{,}440\sqrt{f}}\right)$$

It gives $f$ = 0.02328. Then the pressure drop and the required power input become

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.02328\frac{1\ \text{ft}}{0.75/12\ \text{ft}}\frac{(62.36\ \text{lbm/ft}^3)(6.272\ \text{ft/s})^2}{2}\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}}\right) = 14.2\ \text{lbf/ft}^2$$

$$\dot{W}_{\text{pump}} = \dot{V}\Delta P = \frac{\dot{m}\Delta P}{\rho} = \frac{(1.2\ \text{lbm/s})(14.2\ \text{lbf/ft}^2)}{62.36\ \text{lbm/ft}^3}\left(\frac{1\ \text{W}}{0.737\ \text{lbf}\cdot\text{ft/s}}\right) = \mathbf{0.37\ W}\ \text{(per ft length)}$$

Therefore, useful power input in the amount of 0.37 W is needed per ft of tube length to overcome the frictional losses in the pipe.

***Discussion*** The friction factor could also be determined easily from the explicit Haaland relation. It would give $f$ = 0.02305, which is sufficiently close to 0.02328. Also, the friction factor corresponding to ε = 0 in this case is 0.02306, which indicates that copper pipes can be assumed to be smooth with a negligible error. Also, the power input determined is the mechanical power that needs to be imparted to the fluid. The shaft power will be more than this due to pump inefficiency; the electrical power input will be even more due to motor inefficiency.

**8-43**

**Solution** The pressure of oil in a pipe which discharges into the atmosphere is measured at a certain location. The flow rates are to be determined for 3 different orientations.

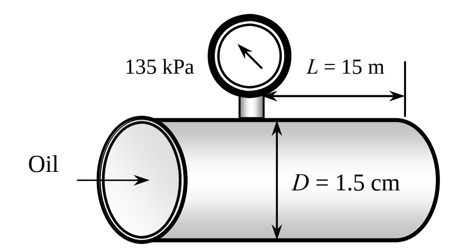
***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The flow is laminar (to be verified). **4** The pipe involves no components such as bends, valves, and connectors. **5** The piping section involves no work devices such as pumps and turbines.

***Properties*** The density and dynamic viscosity of oil are given to be $\rho = 876$ kg/m$^3$ and $\mu = 0.24$ kg/m·s.

***Analysis*** The pressure drop across the pipe and the cross-sectional area are

$$\Delta P = P_1 - P_2 = 135 - 88 = 47 \text{ kPa}$$

$$A_c = \pi D^2 / 4 = \pi (0.015 \text{ m})^2 / 4 = 1.767 \times 10^{-4} \text{ m}^2$$




(*a*) The flow rate for all three cases can be determined from,

$$\dot{V} = \frac{(\Delta P - \rho g L \sin\theta)\pi D^4}{128\mu L}$$

where θ is the angle the pipe makes with the horizontal. For the horizontal case, θ = 0 and thus sin θ = 0. Therefore,

$$\dot{V}_{\text{horiz}} = \frac{\Delta P \pi D^4}{128\mu L} = \frac{(47 \text{ kPa})\pi(0.015 \text{ m})^4}{128(0.24 \text{ kg/m}\cdot\text{s})(15 \text{ m})}\left(\frac{1 \text{ kg}\cdot\text{m/s}^2}{1 \text{ N}}\right)\left(\frac{1000 \text{ N/m}^2}{1 \text{ kPa}}\right) = \mathbf{1.62 \times 10^{-5}\ m^3/s}$$

(*b*) For uphill flow with an inclination of 8°, we have θ = +8°, and

$$\dot{V}_{\text{uphill}} = \frac{(\Delta P - \rho g L \sin\theta)\pi D^4}{128\mu L}$$

$$= \frac{[(47{,}000 \text{ Pa} - (876 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(15 \text{ m})\sin 8°]\pi(0.015 \text{ m})^4}{128(0.24 \text{ kg/m}\cdot\text{s})(15 \text{ m})}\left(\frac{1 \text{ kg}\cdot\text{m/s}^2}{1 \text{ Pa}\cdot\text{m}^2}\right)$$

$$= \mathbf{1.00 \times 10^{-5}\ m^3/s}$$

(*c*) For downhill flow with an inclination of 8°, we have θ = -8°, and

$$\dot{V}_{\text{downhill}} = \frac{(\Delta P - \rho g L \sin\theta)\pi D^4}{128\mu L}$$

$$= \frac{[(47{,}000 \text{ Pa} - (876 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(15 \text{ m})\sin(-8°)]\pi(0.015 \text{ m})^4}{128(0.24 \text{ kg/m}\cdot\text{s})(15 \text{ m})}\left(\frac{1 \text{ kg}\cdot\text{m/s}^2}{1 \text{ Pa}\cdot\text{m}^2}\right)$$

$$= \mathbf{2.24 \times 10^{-5}\ m^3/s}$$

The flow rate is the highest for downhill flow case, as expected. The average fluid velocity and the Reynolds number in this case are

$$V = \frac{\dot{V}}{A_c} = \frac{2.24 \times 10^{-5} \text{.m}^3\text{/s}}{1.767 \times 10^{-4} \text{ m}^2} = 0.127 \text{ m/s}$$

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(876 \text{ kg/m}^3)(0.127 \text{ m/s})(0.015 \text{ m})}{0.24 \text{ kg/m}\cdot\text{s}} = 7.0$$

which is less than 2300. Therefore, the flow is **laminar** for all three cases, and the analysis above is valid.

***Discussion*** Note that the flow is driven by the combined effect of pressure difference and gravity. As can be seen from the calculated rates above, gravity opposes uphill flow, but helps downhill flow. Gravity has no effect on the flow rate in the horizontal case.

**8-44**

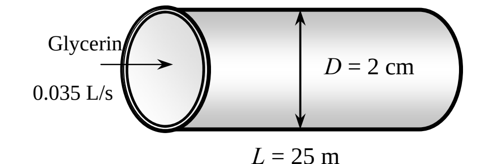
**Solution** Glycerin is flowing through a horizontal pipe which discharges into the atmosphere at a specified flow rate. The absolute pressure at a specified location in the pipe, and the angle $\theta$ that the pipe must be inclined downwards for the pressure in the entire pipe to be atmospheric pressure are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The flow is laminar (to be verified). **4** The pipe involves no components such as bends, valves, and connectors. **5** The piping section involves no work devices such as pumps and turbines.

***Properties*** The density and dynamic viscosity of glycerin at 40°C are given to be $\rho$ = 1252 kg/m$^3$ and $\mu$ = 0.27 kg/m·s.

***Analysis*** (*a*) The flow rate for horizontal or inclined pipe can be determined from

$$\dot{V} = \frac{(\Delta P - \rho g L \sin\theta)\pi D^4}{128\mu L} \quad (1)$$




where θ is the angle the pipe makes with the horizontal. For the horizontal case, θ = 0 and thus sin θ = 0. Therefore,

$$\dot{V}_{\text{horiz}} = \frac{\Delta P \pi D^4}{128\mu L} \quad (2)$$

Solving for $\Delta P$ and substituting,

$$\Delta P = \frac{128\mu L \dot{V}_{\text{horiz}}}{\pi D^4} = \frac{128(0.27 \text{ kg/m}\cdot\text{s})(25\text{ m})(0.035\times10^{-3}\text{ m}^3/\text{s})}{\pi(0.02\text{ m})^4}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)$$

$$= 60.2\text{ kN/m}^2 = 60.2\text{ kPa}$$

Then the pressure 25 m before the pipe exit becomes

$$\Delta P = P_1 - P_2 \quad \rightarrow \quad P_1 = P_2 + \Delta P = 100 + 60.2 = \mathbf{160.2\ kPa}$$

(*b*) When the flow is gravity driven downhill with an inclination θ, and the pressure in the entire pipe is constant at the atmospheric pressure, the hydrostatic pressure rise with depth is equal to pressure drop along the pipe due to frictional effects. Setting $\Delta P = P_1 - P_2 = 0$ in Eq. (1) and substituting, $\theta$ is determined to be

$$\dot{V}_{\text{downhill}} = \frac{\rho g \sin\theta \pi D^4}{128\mu}$$

$$0.035\times10^{-3}\text{ m}^3/\text{s} = \frac{-(1252\text{ kg/m}^3)(9.81\text{ m/s}^2)\sin\theta\pi(0.02\text{ m})^4}{128(0.27\text{ kg/m}\cdot\text{s})} \quad \rightarrow \quad \theta = \mathbf{-11.3°}$$

Therefore, the pipe must be inclined 11.3° downwards from the horizontal to maintain flow in the pipe at the same rate.

***Verification:*** The average fluid velocity and the Reynolds number in this case are

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.035\times10^{-3}\text{ m}^3/\text{s}}{\pi(0.02\text{ m})^2/4} = 0.111\text{ m/s}$$

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(1252\text{ kg/m}^3)(0.111\text{ m/s})(0.02\text{ m})}{0.27\text{ kg/m}\cdot\text{s}} = 10.3$$

which is less than 2300. Therefore, the flow is **laminar**, as assumed, and the analysis above is valid.

***Discussion*** Note that the flow is driven by the combined effect of pressure difference and gravity. Gravity has no effect on the flow rate in the horizontal case, but it governs the flow alone when there is no pressure difference across the pipe.

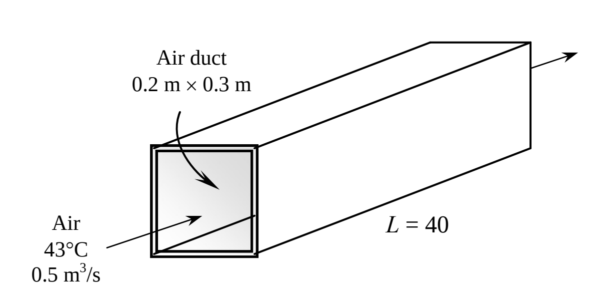
**8-45**

**Solution** Air in a heating system is distributed through a rectangular duct made of commercial steel at a specified rate. The pressure drop and head loss through a section of the duct are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** Air is an ideal gas. **4** The duct involves no components such as bends, valves, and connectors. **5** The flow section involves no work devices such as fans or turbines.

***Properties*** The roughness of commercial steel surfaces is $\varepsilon$ = 0.000045 m. The dynamic viscosity of air at 40°C is $\mu$ = $1.918\times10^{-5}$ kg/m·s, and it is independent of pressure. The density of air listed in that table is for 1 atm. The density at 105 kPa and 315 K can be determined from the ideal gas relation to be

$$\rho = \frac{P}{RT} = \frac{105 \text{ kPa}}{(0.287 \text{ kPa}\cdot\text{m}^3/\text{kg.K})(40+273\text{ K})} = 1.169 \text{ kg/m}^3$$




***Analysis*** The hydraulic diameter, average velocity, and Reynolds number are

$$D_h = \frac{4A_c}{p} = \frac{4ab}{2(a+b)} = \frac{4(0.3\text{ m})(0.20\text{ m})}{2(0.3+0.20)\text{ m}} = 0.24 \text{ m}$$

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{a\times b} = \frac{0.5\text{m}^3/\text{s}}{(0.3\text{ m})(0.2\text{ m})} = 8.333 \text{ m/s}$$

$$\text{Re} = \frac{\rho V D_h}{\mu} = \frac{(1.169 \text{ kg/m}^3)(8.333\text{ m/s})(0.24\text{ m})}{1.918\times10^{-5}\text{ kg/m}\cdot\text{s}} = 121{,}900$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the duct is

$$\varepsilon / D_h = \frac{4.5\times10^{-5}\text{ m}}{0.24\text{ m}} = 1.875\times10^{-4}$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D_h}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{1.875\times10^{-4}}{3.7} + \frac{2.51}{121{,}900\sqrt{f}}\right)$$

It gives $f$ = 0.01833. Then the pressure drop in the duct and the head loss become

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.01833\frac{40\text{ m}}{0.24\text{ m}}\frac{(1.169\text{ kg/m}^3)(8.333\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 124 \text{ N/m}^2 = \mathbf{124\ Pa}$$

$$h_L = \frac{\Delta P_L}{\rho g} = f\frac{L}{D}\frac{V^2}{2g} = 0.01833\frac{40\text{ m}}{0.24\text{ m}}\frac{(8.333\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = \mathbf{10.8\ m}$$

***Discussion*** The required pumping power in this case is

$$\dot{W}_{\text{pump}} = \dot{V}\Delta P = (0.5\text{ m}^3/\text{s})(124\text{ Pa})\left(\frac{1\text{ W}}{1\text{ Pa}\cdot\text{m}^3/\text{s}}\right) = 62 \text{ W}$$

Therefore, 62 W of mechanical power needs to be imparted to the fluid. The shaft power will be more than this due to fan inefficiency; the electrical power input will be even more due to motor inefficiency. Also, the friction factor could be determined easily from the explicit Haaland relation. It would give $f$ = 0.0181, which is sufficiently close to 0.0183.

---

**8-46**

**Solution** Glycerin is flowing through a smooth pipe with a specified average velocity. The pressure drop per 10 m of the pipe is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The pipe involves no components such as bends, valves, and connectors. **4** The piping section involves no work devices such as pumps and turbines.

***Properties*** The density and dynamic viscosity of glycerin at 40°C are given to be $\rho$ = 1252 kg/m$^3$ and $\mu$ = 0.27 kg/m·s, respectively.

***Analysis*** The volume flow rate and the Reynolds number are

$$\dot{V} = VA_c = V(\pi D^2/4) = (3.5\text{ m/s})[\pi(0.05\text{ m})^2/4] = 0.006872\text{ m}^3/\text{s}$$

$$\text{Re} = \frac{\rho VD_h}{\mu} = \frac{(1252\text{ kg/m}^3)(3.5\text{ m/s})(0.05\text{ m})}{0.27\text{ kg/m}\cdot\text{s}} = 811.5$$

Glycerin
3.5 m/s
$D$ = 5 cm
$L$ = 10 m

which is less than 2300. Therefore, the flow is laminar, and the friction factor for this circular pipe is

$$f = \frac{64}{\text{Re}} = \frac{64}{811.5} = 0.07887$$

Then the pressure drop in the pipe becomes

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.07887\frac{10\text{ m}}{0.05\text{ m}}\frac{(1252\text{ kg/m}^3)(3.5\text{ m/s})^2}{2}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right) = \mathbf{121\text{ kPa}}$$

***Discussion*** The required pumping power in this case is

$$\dot{W}_{\text{pump}} = \dot{V}\Delta P = (0.006872\text{ m}^3/\text{s})(121\text{ kPa})\left(\frac{1\text{ kW}}{1\text{ kPa}\cdot\text{m}^3/\text{s}}\right) = \mathbf{0.83\text{ kW}}$$

Therefore, 0.83 kW of mechanical power needs to be imparted to the fluid. The shaft power will be more than this due to pump inefficiency; the electrical power input will be even more due to motor inefficiency.



**8-47**

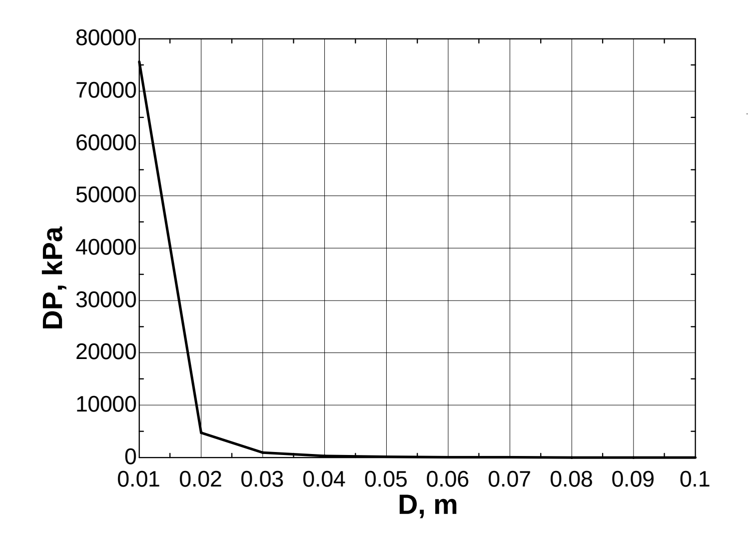


**Solution** In the previous problem, the effect of the pipe diameter on the pressure drop for the same constant flow rate is to be investigated by varying the pipe diameter from 1 cm to 10 cm in increments of 1 cm.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
g=9.81
Vdot=3.5*pi*(0.05)^2/4
Ac=pi*D^2/4
rho= 1252
nu=mu/rho
mu=0.27
L= 10
V=Vdot/Ac
"Reynolds number"
Re=V*D/nu
f=64/Re
DP=f*(L/D)*rho*V^2/2000 "kPa"
W=Vdot*DP "kW"
```

| $D$, m | $\Delta P$, kPa | $V$, m/s | Re |
|---|---|---|---|
| 0.01 | 75600 | 87.5 | 4057 |
| 0.02 | 4725 | 21.88 | 2029 |
| 0.03 | 933.3 | 9.722 | 1352 |
| 0.04 | 295.3 | 5.469 | 1014 |
| 0.05 | 121 | 3.5 | 811.5 |
| 0.06 | 58.33 | 2.431 | 676.2 |
| 0.07 | 31.49 | 1.786 | 579.6 |
| 0.08 | 18.46 | 1.367 | 507.2 |
| 0.09 | 11.52 | 1.08 | 450.8 |
| 0.1 | 7.56 | 0.875 | 405.7 |




***Discussion*** The pressure drop decays quite rapidly with increasing diameter – by several orders of magnitude, in fact. We conclude that larger diameter pipes are better when pressure drop is of concern. Of course, bigger pipes cost more and take up more space, so there is typically an optimum pipe size that is a compromise between cost and practicality.

**8-48E**

**Solution** Air is flowing through a square duct made of commercial steel at a specified rate. The pressure drop and head loss per ft of duct are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** Air is an ideal gas. **4** The duct involves no components such as bends, valves, and connectors. **5** The flow section involves no work devices such as fans or turbines.

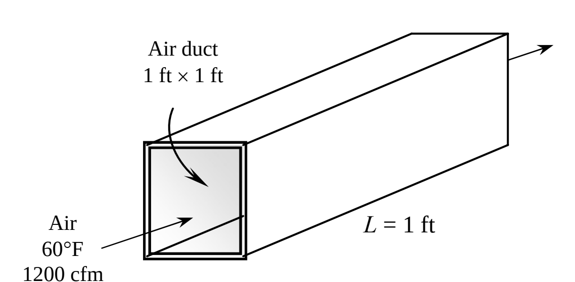
***Properties*** The density and dynamic viscosity of air at 1 atm and 60°F are $\rho$ = 0.07633 lbm/ft$^3$, $\mu$ = 0.04365 lbm/ft·h, and $\nu$ = 0.5718 ft$^2$/s = 1.588×10$^{-4}$ ft$^2$/s. The roughness of commercial steel surfaces is $\varepsilon$ = 0.00015 ft.

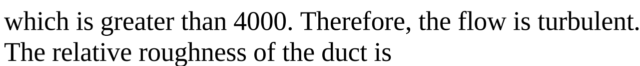
***Analysis*** The hydraulic diameter, the average velocity, and the Reynolds number in this case are

$$D_h = \frac{4A_c}{p} = \frac{4a^2}{4a} = a = 1\text{ ft}$$

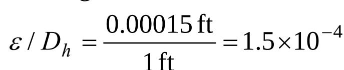
$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{a^2} = \frac{1200\text{ ft}^3\text{/min}}{(1\text{ ft})^2} = 1200\text{ ft/min} = 20\text{ ft/s}$$

$$\text{Re} = \frac{VD_h}{\nu} = \frac{(20\text{ ft/s})(1\text{ ft})}{1.588\times10^{-4}\text{ ft}^2\text{/s}} = 1.259\times10^5$$

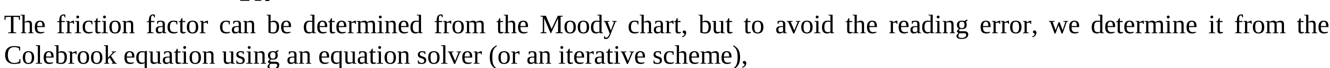


which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the duct is



$$\varepsilon / D_h = \frac{0.00015\text{ ft}}{1\text{ ft}} = 1.5\times10^{-4}$$



The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon / D_h}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{1.5\times10^{-4}}{3.7} + \frac{2.51}{125{,}900\sqrt{f}}\right)$$

It gives $f$ = 0.0180. Then the pressure drop in the duct and the head loss become

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.0180\frac{1\text{ ft}}{1\text{ ft}}\frac{(0.07633\text{ lbm/ft}^3)(20\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}}\right) = \mathbf{8.53\times10^{-3}\ lbf/ft^2}$$

$$h_L = \frac{\Delta P_L}{\rho g} = f\frac{L}{D}\frac{V^2}{2g} = 0.0180\frac{1\text{ ft}}{1\text{ ft}}\frac{(20\text{ ft/s})^2}{2(32.2\text{ ft/s}^2)} = \mathbf{0.112\ ft}$$

***Discussion*** The required pumping power in this case is

$$\dot{W}_{\text{pump}} = \dot{V}\Delta P = (1200/60\text{ ft}^3\text{/s})(8.53\times10^{-3}\text{ lbf/ft}^2)\left(\frac{1\text{ W}}{0.737\text{ lbf}\cdot\text{ft/s}}\right) = 0.231\text{ W (per ft length)}$$

Therefore, 0.231 W of mechanical power needs to be imparted to the fluid per ft length of the duct. The shaft power will be more than this due to fan inefficiency; the electrical power input will be even more due to motor inefficiency. Also, the friction factor could be determined easily from the explicit Haaland relation. It would give $f$ = 0.0178, which is sufficiently close to 0.0180.

---

**8-49**

**Solution** Liquid ammonia is flowing through a copper tube at a specified mass flow rate. The pressure drop, the head loss, and the pumping power required to overcome the frictional losses in the tube are to be determined.

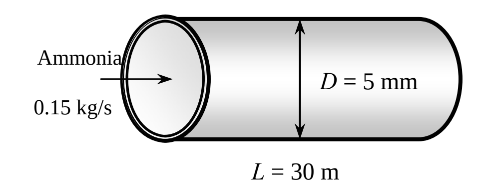
***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The pipe involves no components such as bends, valves, and connectors. **4** The piping section involves no work devices such as pumps and turbines.

***Properties*** The density and dynamic viscosity of liquid ammonia at -20°C are $\rho$ = 665.1 kg/m$^3$ and $\mu$ = 2.361×10$^{-4}$ kg/m·s. The roughness of copper tubing is 1.5×10$^{-6}$ m.

***Analysis*** First we calculate the average velocity and the Reynolds number to determine the flow regime:

$$V = \frac{\dot m}{\rho A_c} = \frac{\dot m}{\rho(\pi D^2/4)} = \frac{0.15\ \text{kg/s}}{(665.1\ \text{kg/m}^3)[\pi(0.005\ \text{m})^2/4]} = 11.49\ \text{m/s}$$

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(665.1\ \text{kg/m}^3)(11.49\ \text{m/s})(0.005\ \text{m})}{2.361\times10^{-4}\ \text{kg/m}\cdot\text{s}} = 1.618\times10^5$$




which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D = \frac{1.5\times10^{-6}\ \text{m}}{0.005\ \text{m}} = 3\times10^{-4}$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{3\times10^{-4}}{3.7} + \frac{2.51}{1.618\times10^5\sqrt{f}}\right)$$

It gives $f$ = 0.01819. Then the pressure drop, the head loss, and the useful pumping power required become

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2}$$

$$= 0.01819\frac{30\ \text{m}}{0.005\ \text{m}}\frac{(665.1\ \text{kg/m}^3)(11.49\ \text{m/s})^2}{2}\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1\ \text{kN/m}^2}\right) = 4792\ \text{kPa} \cong \mathbf{4790\ kPa}$$

$$h_L = \frac{\Delta P_L}{\rho g} = f\frac{L}{D}\frac{V^2}{2g} = 0.01819\frac{30\ \text{m}}{0.005\ \text{m}}\frac{(11.49\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)} = \mathbf{734\ m}$$

$$\dot W_{\text{pump}} = \dot V\Delta P = \frac{\dot m\Delta P}{\rho} = \frac{(0.15\ \text{kg/s})(4792\ \text{kPa})}{665.1\ \text{kg/m}^3}\left(\frac{1\ \text{kW}}{1\ \text{kPa}\cdot\text{m}^3/\text{s}}\right) = \mathbf{1.08\ kW}$$

Therefore, useful power input in the amount of 1.08 kW is needed to overcome the frictional losses in the tube.

***Discussion*** The friction factor could also be determined easily from the explicit Haaland relation. It would give $f$ = 0.0180, which is sufficiently close to 0.0182. The friction factor corresponding to ε = 0 in this case is 0.0163, which is about 10% lower. Therefore, the copper tubes in this case are nearly "smooth".

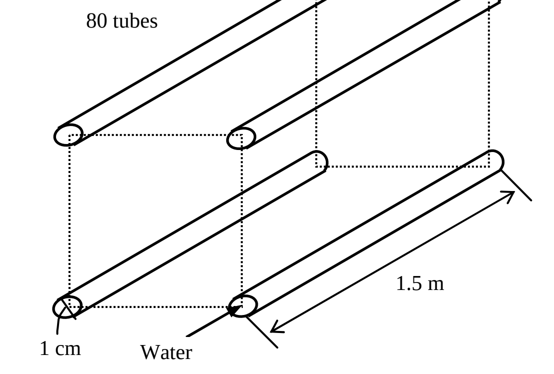
Also, the power input determined is the mechanical power that needs to be imparted to the fluid. The shaft power will be more than this due to pump inefficiency; the electrical power input will be even more due to motor inefficiency.

**8-50** **[*Also solved using EES on enclosed DVD*]**

**Solution** Water is flowing through a brass tube bank of a heat exchanger at a specified flow rate. The pressure drop and the pumping power required are to be determined. Also, the percent reduction in the flow rate of water through the tubes is to be determined after scale build-up on the inner surfaces of the tubes.

***Assumptions*** **1** The flow is steady, horizontal, and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed (this is a questionable assumption since the tubes are short, and it will be verified). **3** The inlet, exit, and header losses are negligible, and the tubes involve no components such as bends, valves, and connectors. **4** The piping section involves no work devices such as pumps and turbines.

***Properties*** The density and dynamic viscosity of water at 20°C are $\rho$ = 983.3 kg/m$^3$ and $\mu$ = 0.467×10$^{-3}$ kg/m·s, respectively. The roughness of brass tubing is 1.5×10$^{-6}$ m.




***Analysis*** First we calculate the average velocity and the Reynolds number to determine the flow regime:

$$V = \frac{\dot V}{A_c} = \frac{\dot V}{N_{\text{tune}}(\pi D^2/4)} = \frac{0.015\ \text{m}^3/\text{s}}{80[\pi(0.01\ \text{m})^2/4]} = 2.387\ \text{m/s}$$

$$\text{Re} = \frac{\rho V D_h}{\mu} = \frac{(983.3\ \text{kg/m}^3)(2.387\ \text{m/s})(0.01\ \text{m})}{0.467\times10^{-3}\ \text{kg/m}\cdot\text{s}} = 50{,}270$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D = \frac{1.5\times10^{-6}\ \text{m}}{0.01\ \text{m}} = 1.5\times10^{-4}$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt f} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{\text{Re}\sqrt f}\right) \rightarrow \frac{1}{\sqrt f} = -2.0\log\left(\frac{1.5\times10^{-4}}{3.7} + \frac{2.51}{50{,}270\sqrt f}\right)$$

It gives $f$ = 0.0214. Then the pressure drop, the head loss, and the useful pumping power required become

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.0214\frac{1.5\ \text{m}}{0.01\ \text{m}}\frac{(983.3\ \text{kg/m}^3)(2.387\ \text{m/s})^2}{2}\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1\ \text{kN/m}^2}\right) = \mathbf{8.99\ kPa}$$

$$\dot W_{\text{pump}} = \dot V\Delta P = (0.015\ \text{m}^3/\text{s})(8.99\ \text{kPa})\left(\frac{1\ \text{kW}}{1\ \text{kPa}\cdot\text{m}^3/\text{s}}\right) = \mathbf{0.135\ kW}$$

Therefore, useful power input in the amount of 0.135 kW is needed to overcome the frictional losses in the tube. The hydrodynamic entry length in this case is

$$L_{\text{h,turbulent}} \approx 10D = 10(0.01\ \text{m}) = 0.1\ \text{m}$$

which is much less than 1.5 m. Therefore, the assumption of fully developed flow is valid. (The effect of the entry region is to increase the friction factor, and thus the pressure drop and pumping power).

**After scale buildup**: When 1-mm thick scale builds up on the inner surfaces (and thus the diameter is reduced to 0.8 cm from 1 cm) with an equivalent roughness of 0.4 mm, and the useful power input is fixed at 0.135 kW, the problem can be formulated as follows (note that the flow rate and thus the average velocity are unknown in this case):

$$V = \frac{\dot V}{A_c} = \frac{\dot V}{N_{\text{tune}}(\pi D^2/4)} \rightarrow V = \frac{\dot V}{80[\pi(0.008\ \text{m})^2/4]} \quad (1)$$

$$\text{Re} = \frac{\rho V D}{\mu} \rightarrow \text{Re} = \frac{(983.3\ \text{kg/m}^3)V(0.008\ \text{m})}{0.467\times10^{-3}\ \text{kg/m}\cdot\text{s}} \quad (2)$$

$$\varepsilon/D = \frac{0.0004\ \text{m}}{0.008\ \text{m}} = 0.05$$

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon / D}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{0.05}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \qquad (3)$$

$$\Delta P = f\frac{L}{D}\frac{\rho V^2}{2} \rightarrow \Delta P = f\frac{1.5\text{ m}}{0.008\text{ m}}\frac{(983.3\text{ kg/m}^3)V^2}{2}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right) \qquad (4)$$

$$\dot{W}_{\text{pump}} = 0.135\text{ kW} \rightarrow \dot{V}\Delta P = 0.135 \qquad (5)$$

Solving this system of 5 equations in 5 unknown ($f$, Re, $V$, $\Delta P$, and $\dot{V}$) using an equation solver (or a trial-and-error approach, by assuming a velocity value) gives

$f = 0.0723$, Re = 28,870, $V = 1.714$ m/s, $\Delta P = 19.6$ kPa, and $\dot{V} = 0.00689$ m$^3$/s = 6.89 L/s

Then the percent reduction in the flow rate becomes

$$\text{Reduction ratio} = \frac{\dot{V}_{\text{clean}} - \dot{V}_{\text{dirty}}}{\dot{V}_{\text{clean}}} = \frac{15-6.89}{15} = 0.54 = \mathbf{54\%}$$

Therefore, for the same pump input, the flow rate will be reduced to less than half of the original flow rate when the pipes were new and clean.

***Discussion*** The friction factor could also be determined easily from the explicit Haaland relation. Also, the power input determined is the mechanical power that needs to be imparted to the fluid. The shaft power will be more than this due to pump inefficiency; the electrical power input will be even more due to motor inefficiency.

---

## Minor Losses

---

### 8-51C

**Solution** We are to define minor loss and minor loss coefficient.

***Analysis*** The **head losses associated with the flow of a fluid through fittings, valves, bends, elbows, tees, inlets, exits, enlargements, contractions, etc.** are called *minor losses*, and are expressed in terms of the *minor loss coefficient* as

$$K_L = \frac{h_L}{V^2/(2g)}$$

***Discussion*** Basically, any irreversible loss that is not due to friction in long, straight sections of pipe is a minor loss.

---

### 8-52C

**Solution** We are to define equivalent length and its relationship to the minor loss coefficient.

***Analysis*** *Equivalent length* is **the length of a straight pipe which would give the same head loss as the minor loss component**. It is related to the minor loss coefficient by

$$L_{\text{equiv}} = \frac{D}{f}K_L$$

***Discussion*** Equivalent length is not as universal as minor loss coefficient because it depends on the roughness and Reynolds number of the equivalent straight section of pipe.

---

**8-53C**

**Solution** We are to discuss the effect of rounding a pipe inlet.

***Analysis*** The effect of rounding of a pipe inlet on the loss coefficient is **(*c*) very significant**.

***Discussion*** In fact, the minor loss coefficient changes from 0.8 for a reentrant pipe inlet to about 0.03 for a well-rounded pipe inlet – quite a significant improvement.

---

**8-54C**

**Solution** We are to discuss the effect of rounding on a pipe outlet.

***Analysis*** The effect of rounding of a pipe exit on the loss coefficient is **(*a*) negligible**.

***Discussion*** At any pipe outlet, all the kinetic energy is wasted, and the minor loss coefficient is equal to $\alpha$, which is about 1.05 for fully developed turbulent pipe flow. Rounding of the outlet does not help.

---

**8-55C**

**Solution** We are to compare the minor losses of a gradual expansion and a gradual contraction.

***Analysis*** **A gradual expansion, in general, has a greater minor loss coefficient than a gradual contraction** in pipe flow. This is due to the adverse pressure gradient in the boundary layer, which may lead to flow separation.

***Discussion*** Note, however, that pressure is "recovered" in a gradual expansion. In other words, the pressure rises in the direction of flow. Such a device is called a *diffuser*.

---

**8-56C**

**Solution** We are to discuss ways to reduce the head loss in a pipe flow with bends.

***Analysis*** Another way of reducing the head loss associated with turns is to **install turning vanes inside the elbows**.

***Discussion*** There are many other possible answers, such as: **reduce the inside wall roughness of the pipe**, **use a larger diameter pipe**, **shorten the length of pipe as much as possible**, etc.

---

**8-57C**

**Solution** We are to compare two different ways to reduce the minor loss in pipe bends.

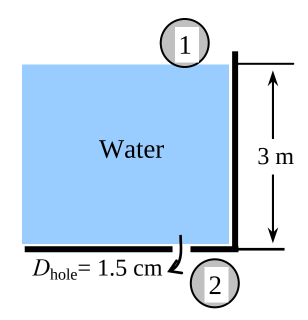
***Analysis*** The loss coefficient is lower for flow through a 90° miter elbow with well-designed vanes ($K_L \approx 0.2$) than it is for flow through a smooth curved bend ($K_L \approx 0.9$). Therefore, **using miter elbows with vanes results in a greater reduction in pumping power requirements**.

***Discussion*** Both values are for threaded elbows. The loss coefficients for flanged elbows are much lower.

---

**8-58**

**Solution** Water is to be withdrawn from a water reservoir by drilling a hole at the bottom surface. The flow rate of water through the hole is to be determined for the well-rounded and sharp-edged entrance cases.




***Assumptions*** **1** The flow is steady and incompressible. **2** The reservoir is open to the atmosphere so that the pressure is atmospheric pressure at the free surface. **3** The effect of the kinetic energy correction factor is disregarded, and thus $\alpha = 1$.

***Analysis*** The loss coefficient is $K_L = 0.5$ for the sharp-edged entrance, and $K_L = 0.03$ for the well-rounded entrance. We take point 1 at the free surface of the reservoir and point 2 at the exit of the hole. We also take the reference level at the exit of the hole ($z_2 = 0$). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is zero ($V_1 = 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump,u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad z_1=\alpha_2\frac{V_2^2}{2g}+h_L$$

where the head loss is expressed as $h_L = K_L\dfrac{V^2}{2g}$. Substituting and solving for $V_2$ gives

$$z_1=\alpha_2\frac{V_2^2}{2g}+K_L\frac{V_2^2}{2g} \quad\rightarrow\quad 2gz_1=V_2^2(\alpha_2+K_L) \quad\rightarrow\quad V_2=\sqrt{\frac{2gz_1}{\alpha_2+K_L}}=\sqrt{\frac{2gz_1}{1+K_L}}$$

since $\alpha_2 = 1$. Note that in the special case of $K_L = 0$, it reduces to the Toricelli equation $V_2=\sqrt{2gz_1}$, as expected. Then the volume flow rate becomes

$$\dot{V}=A_cV_2=\frac{\pi D_{hole}^2}{4}\sqrt{\frac{2gz_1}{1+K_L}}$$

Substituting the numerical values, the flow rate for both cases are determined to be

*Well-rounded entrance*: $\dot{V}=\dfrac{\pi D_{hole}^2}{4}\sqrt{\dfrac{2gz_1}{1+K_L}}=\dfrac{\pi(0.015\text{ m})^2}{4}\sqrt{\dfrac{2(9.81\text{ m/s}^2)(3\text{ m})}{1+0.03}}=\mathbf{1.34\times10^{-3}\ m^3/s}$

*Sharp-edged entrance*: $\dot{V}=\dfrac{\pi D_{hole}^2}{4}\sqrt{\dfrac{2gz_1}{1+K_L}}=\dfrac{\pi(0.015\text{m})^2}{4}\sqrt{\dfrac{2(9.81\text{ m/s}^2)(3\text{ m})}{1+0.5}}=\mathbf{1.11\times10^{-3}\ m^3/s}$

***Discussion*** The flow rate in the case of frictionless flow ($K_L = 0$) is $1.36\times10^{-3}$ m$^3$/s. Note that the frictional losses cause the flow rate to decrease by 1.5% for well-rounded entrance, and 18.5% for the sharp-edged entrance.

**8-59**

**Solution** Water is discharged from a water reservoir through a circular hole of diameter $D$ at the side wall at a vertical distance $H$ from the free surface. A relation for the "equivalent diameter" of the sharp-edged hole for use in frictionless flow relations is to be obtained.

***Assumptions*** **1** The flow is steady and incompressible. **2** The reservoir is open to the atmosphere so that the pressure is atmospheric pressure at the free surface. **3** The effect of the kinetic energy correction factor is disregarded, and thus $\alpha = 1$.

***Analysis*** The loss coefficient is $K_L = 0.5$ for the sharp-edged entrance, and $K_L = 0$ for the "frictionless" flow. We take point 1 at the free surface of the reservoir and point 2 at the exit of the hole, which is also taken to be the reference level ($z_2 = 0$). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is zero ($V_1 = 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump,u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbin e, e}} + h_L \quad \rightarrow \quad H = \alpha_2 \frac{V_2^2}{2g} + h_L$$

where the head loss is expressed as $h_L = K_L \frac{V^2}{2g}$. Substituting and solving for $V_2$ gives

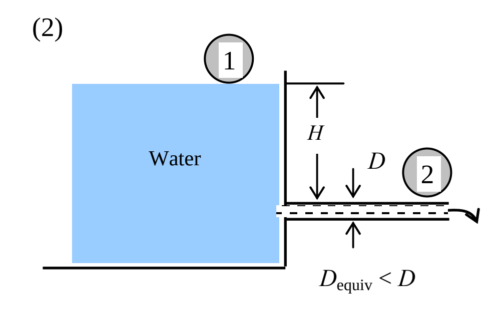
$$H = \alpha_2 \frac{V_2^2}{2g} + K_L \frac{V_2^2}{2g} \quad \rightarrow \quad 2gH = V_2^2(\alpha_2 + K_L) \quad \rightarrow \quad V_2 = \sqrt{\frac{2gH}{\alpha_2 + K_L}} = \sqrt{\frac{2gH}{1 + K_L}}$$

since $\alpha_2 = 1$. Then the volume flow rate becomes

$$\dot{V} = A_c V_2 = \frac{\pi D^2}{4} \sqrt{\frac{2gH}{1 + K_L}} \qquad (1)$$

Note that in the special case of $K_L = 0$ (frictionless flow), the velocity relation reduces to the Toricelli equation, $V_{2,\text{frictionless}} = \sqrt{2gH}$ . The flow rate in this case through a hole of $D_e$ (equivalent diameter) is

$$\dot{V} = A_{c,\text{equiv}} V_{2,\text{frictionless}} = \frac{\pi D_{\text{equiv}}^2}{4} \sqrt{2gH} \qquad (2)$$




Setting Eqs. (1) and (2) equal to each other gives the desired relation for the equivalent diameter,

$$\frac{\pi D_{\text{equiv}}^2}{4} \sqrt{2gH} = \frac{\pi D^2}{4} \sqrt{\frac{2gH}{1 + K_L}}$$

which gives

$$D_{\text{equiv}} = \frac{D}{(1 + K_L)^{1/4}} = \frac{D}{(1 + 0.5)^{1/4}} = \mathbf{0.904\, D}$$

***Discussion*** Note that the effect of frictional losses of a sharp-edged entrance is to reduce the diameter by about 10%. Also, noting that the flow rate is proportional to the square of the diameter, we have $\dot{V} \propto D_{\text{equiv}}^2 = (0.904D)^2 = 0.82D^2$. Therefore, the flow rate through a sharp-edged entrance is about 18% less compared to the frictionless entrance case.

**8-60**

**Solution** Water is discharged from a water reservoir through a circular hole of diameter $D$ at the side wall at a vertical distance $H$ from the free surface. A relation for the "equivalent diameter" of the slightly rounded hole for use in frictionless flow relations is to be obtained.

***Assumptions*** **1** The flow is steady and incompressible. **2** The reservoir is open to the atmosphere so that the pressure is atmospheric pressure at the free surface. **3** The effect of the kinetic energy correction factor is disregarded, and thus $\alpha = 1$.

***Analysis*** The loss coefficient is $K_L = 0.12$ for the slightly rounded entrance, and $K_L = 0$ for the "frictionless" flow.

We take point 1 at the free surface of the reservoir and point 2 at the exit of the hole, which is also taken to be the reference level ($z_2 = 0$). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is zero ($V_1 = 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump,u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbin e, e}}+h_L \quad\rightarrow\quad H=\alpha_2\frac{V_2^2}{2g}+h_L$$

where the head loss is expressed as $h_L = K_L\dfrac{V^2}{2g}$. Substituting and solving for $V_2$ gives

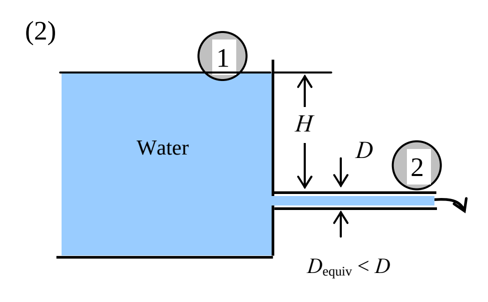
$$H=\alpha_2\frac{V_2^2}{2g}+K_L\frac{V_2^2}{2g} \quad\rightarrow\quad 2gH=V_2^2(\alpha_2+K_L) \quad\rightarrow\quad V_2=\sqrt{\frac{2gH}{\alpha_2+K_L}}=\sqrt{\frac{2gH}{1+K_L}}$$

since $\alpha_2 = 1$. Then the volume flow rate becomes

$$\dot{V}=A_cV_2=\frac{\pi D^2}{4}\sqrt{\frac{2gH}{1+K_L}} \qquad (1)$$

Note that in the special case of $K_L = 0$ (frictionless flow), the velocity relation reduces to the Toricelli equation, $V_{2,\text{frictionless}}=\sqrt{2gz_1}$. The flow rate in this case through a hole of $D_e$ (equivalent diameter) is

$$\dot{V}=A_{c,\text{equiv}}V_{2,\text{frictionless}}=\frac{\pi D_{\text{equiv}}^2}{4}\sqrt{2gH} \qquad (2)$$




Setting Eqs. (1) and (2) equal to each other gives the desired relation for the equivalent diameter,

$$\frac{\pi D_{\text{equiv}}^2}{4}\sqrt{2gH}=\frac{\pi D^2}{4}\sqrt{\frac{2gH}{1+K_L}}$$

which gives

$$D_{\text{equiv}}=\frac{D}{(1+K_L)^{1/4}}=\frac{D}{(1+0.12)^{1/4}}=\mathbf{0.972\,D}$$

***Discussion*** Note that the effect of frictional losses of a slightly rounded entrance is to reduce the diameter by about 3%. Also, noting that the flow rate is proportional to the square of the diameter, we have $\dot{V}\propto D_{\text{equiv}}^2=(0.972D)^2=0.945D^2$. Therefore, the flow rate through a slightly rounded entrance is about 5% less compared to the frictionless entrance case.

**8-61**

**Solution** A horizontal water pipe has an abrupt expansion. The water velocity and pressure in the smaller diameter pipe are given. The pressure after the expansion and the error that would have occurred if the Bernoulli Equation had been used are to be determined.

***Assumptions*** **1** The flow is steady, horizontal, and incompressible. **2** The flow at both the inlet and the outlet is fully developed and turbulent with kinetic energy corrections factors of $\alpha_1 = \alpha_2 = 1.06$ (given).

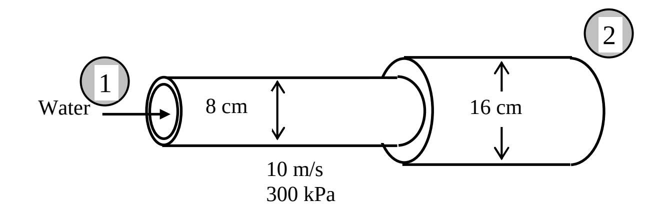
***Properties*** We take the density of water to be $\rho = 1000$ kg/m$^3$.

***Analysis*** Noting that ρ = const. (incompressible flow), the downstream velocity of water is

$$\dot{m}_1 = \dot{m}_2 \;\rightarrow\; \rho V_1 A_1 = \rho V_2 A_2 \;\rightarrow\; V_2 = \frac{A_1}{A_2}V_1 = \frac{\pi D_1^2/4}{\pi D_2^2/4}V_1 = \frac{D_1^2}{D_2^2}V_1 = \frac{(0.08\text{ m})^2}{(0.16\text{ m})^2}(10\text{ m/s}) = 2.5\text{ m/s}$$

The loss coefficient for sudden expansion and the head loss can be calculated from

$$K_L = \left(1-\frac{A_{\text{small}}}{A_{\text{large}}}\right)^2 = \left(1-\frac{D_1^2}{D_2^2}\right)^2 = \left(1-\frac{0.08^2}{0.16^2}\right)^2 = 0.5625$$




$$h_L = K_L\frac{V_1^2}{2g} = (0.5625)\frac{(10\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 2.87\text{ m}$$

Noting that $z_1 = z_2$ and there are no pumps or turbines involved, the energy equation for the expansion section can be expressed in terms of heads as

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump,u}} = \frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad \frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g} = \frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+h_L$$

Solving for $P_2$ and substituting,

$$P_2 = P_1+\rho\left\{\frac{\alpha_1 V_1^2-\alpha_2 V_2^2}{2}-gh_L\right\}$$

$$= (300\text{ kPa})+(1000\text{ kg/m}^3)\left\{\frac{1.06(10\text{ m/s})^2-1.06(2.5\text{ m/s})^2}{2}-(9.81\text{ m/s}^2)(2.87\text{ m})\right\}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right)$$

$$= \mathbf{322\ kPa}$$

Therefore, despite the head (and pressure) loss, the pressure increases from 300 kPa to 321 kPa after the expansion. This is due to the conversion of dynamic pressure to static pressure when the velocity is decreased.

When the head loss is disregarded, the downstream pressure is determined from the Bernoulli equation to be

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1 = \frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad \frac{P_1}{\rho g}+\frac{V_1^2}{2g} = \frac{P_2}{\rho g}+\frac{V_2^2}{2g} \quad\rightarrow\quad P_1 = P_1+\rho\frac{V_1^2-V_2^2}{2}$$

Substituting,

$$P_2 = (300\text{ kPa})+(1000\text{ kg/m}^3)\frac{(10\text{ m/s})^2-(2.5\text{ m/s})^2}{2}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right) = 347\text{ kPa}$$

Therefore, the error in the Bernoulli equation is Error = $P_{2,\text{ Bernoulli}} - P_2$ = 347 – 322 = **25.0 kPa**

Note that the use of the Bernoulli equation results in an error of (347 – 322) / 322 = 0.078 or 7.8%.

***Discussion*** It is common knowledge that higher pressure upstream is necessary to cause flow, and it may come as a surprise that the downstream pressure has *increased* after the abrupt expansion, despite the loss. This is because the sum of the three Bernoulli terms which comprise the total head, consisting of pressure head, velocity head, and elevation head, namely [$P/\rho g$ + ½$V^2/g$+ $z$], drives the flow. With a geometric flow expansion, initially higher velocity head is converted to downstream pressure head, and this increase outweighs the non-convertible and non-recoverable head loss term.

## Piping Systems and Pump Selection

**8-62C**

**Solution** We are to compare the flow rate and pressure drop in two pipes of different diameters in series.

***Analysis*** For a piping system that involves two pipes of different diameters (but of identical length, material, and roughness) connected in *series*, (*a*) **the flow rate through both pipes is the same** and (*b*) **the pressure drop through the smaller diameter pipe is larger**.

***Discussion*** The wall shear stress on the smaller pipe is larger, friction factor *f* is larger, and thus the head loss is higher.

---

**8-63C**

**Solution** We are to compare the flow rate and pressure drop in two pipes of different diameters in parallel.

***Analysis*** For a piping system that involves two pipes of different diameters (but of identical length, material, and roughness) connected in *parallel*, (*a*) **the flow rate through the larger diameter pipe is larger** and (*b*) **the pressure drop through both pipes is the same**.

***Discussion*** Since the two pipes separate from each other but then later re-join, the pressure drop between the two junctions must be the same, regardless of which pipe segment is under consideration.

---

**8-64C**

**Solution** We are to compare the pressure drop of two different-length pipes in parallel.

***Analysis*** **The pressure drop through both pipes is the *same*** since the pressure at a point has a single value, and the inlet and exits of these the pipes connected in parallel coincide.

***Discussion*** The length, diameter, roughness, and number and type of minor losses are all irrelevant – for any two pipes in parallel, both have the same pressure drop.

---

**8-65C**

**Solution** We are to discuss whether the required pump head is equal to the elevation difference when irreversible head losses are negligible.

***Analysis*** **Yes**, when the head loss is negligible, the required pump head is equal to the elevation difference between the free surfaces of the two reservoirs.

***Discussion*** A pump in a piping system may: (1) raise the fluid's elevation, and/or (2) increase the fluid's kinetic energy, and/or (3) increase the fluid's pressure, and/or (4) overcome irreversible losses. In this case, (2), (3), and (4) are zero or negligible; thus only (1) remains.

---

**8-66C**

**Solution** We are to explain how the operating point of a pipe/pump system is determined.

***Analysis*** The pump installed in a piping system **operates at the point where the *system curve* and the *characteristic curve* intersect**. This **point of intersection** is called the *operating point*.

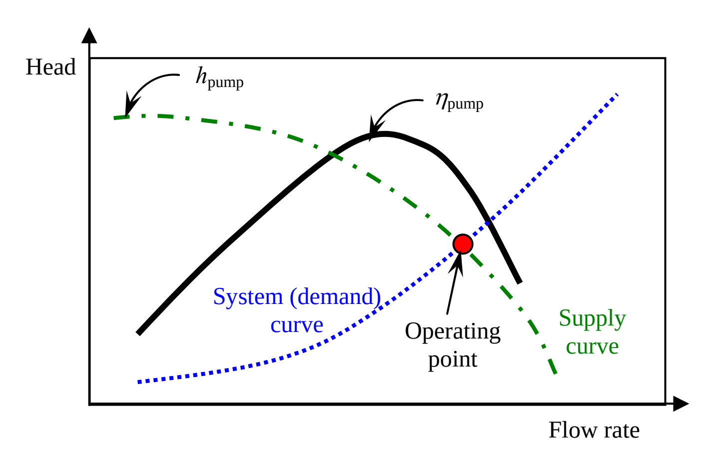
***Discussion*** The volume flow rate "automatically" adjusts itself to reach the operating point.

---

**8-67C**

**Solution** We are to draw a pump head versus flow rate chart and identify several parameters.

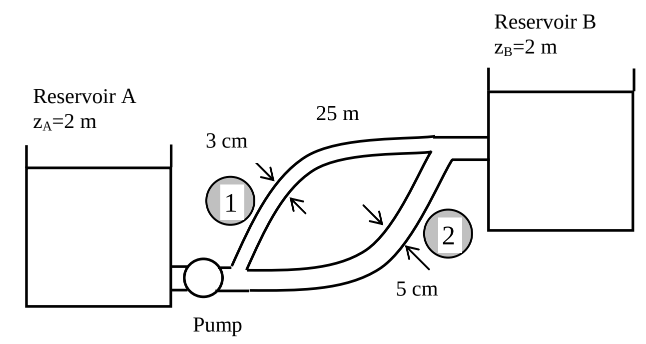
*Analysis* The **plot of the head loss versus the flow rate** is called the *system curve*. The experimentally determined **pump head and pump efficiency versus the flow rate curves** are called *characteristic curves*. **The pump installed in a piping system operates at the point where the *system curve* and the *characteristic curve* intersect**. This point of intersection is called the *operating point*.




*Discussion* By matching the system (demand) curve and the .

**8-68** **[*Also solved using EES on enclosed DVD*]**

**Solution** The pumping power input to a piping system with two parallel pipes between two reservoirs is given. The flow rates are to be determined.




***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The elevations of the reservoirs remain constant. **4** The minor losses and the head loss in pipes other than the parallel pipes are said to be negligible. **5** The flows through both pipes are turbulent (to be verified).

***Properties*** The density and dynamic viscosity of water at 20°C are $\rho$ = 998 kg/m$^3$ and $\mu$ = 1.002×10$^{-3}$ kg/m⋅s. Plastic pipes are smooth, and their roughness is zero, $\varepsilon = 0$.

***Analysis*** This problem cannot be solved directly since the velocities (or flow rates) in the pipes are not known. Therefore, we would normally use a trial-and-error approach here. However, nowadays the equation solvers such as EES are widely available, and thus below we will simply set up the equations to be solved by an equation solver. The head supplied by the pump to the fluid is determined from

$$\dot W_{\text{elect,in}} = \frac{\rho \dot V g h_{\text{pump, u}}}{\eta_{\text{pump-motor}}} \quad\rightarrow\quad 7000\text{ W} = \frac{(998\text{ kg/m}^3)\dot V(9.81\text{ m/s}^2)h_{\text{pump, u}}}{0.68} \qquad (1)$$

We choose points $A$ and $B$ at the free surfaces of the two reservoirs. Noting that the fluid at both points is open to the atmosphere (and thus $P_A = P_B = P_{\text{atm}}$) and that the fluid velocities at both points are zero ($V_A = V_B = 0$), the energy equation for a control volume between these two points simplifies to

$$\frac{P_A}{\rho g} + \alpha_A \frac{V_A^2}{2g} + z_A + h_{\text{pump, u}} = \frac{P_B}{\rho g} + \alpha_B \frac{V_B^2}{2g} + z_B + h_{\text{turbine, e}} + h_L \quad\rightarrow\quad h_{\text{pump, u}} = (z_B - z_A) + h_L$$

or

$$h_{\text{pump, u}} = (9-2) + h_L \quad (2)$$

where

$$h_L = h_{L,1} = h_{L,2} \qquad (3)\,(4)$$

We designate the 3-cm diameter pipe by 1 and the 5-cm diameter pipe by 2. The average velocity, Reynolds number, friction factor, and the head loss in each pipe are expressed as

$$V_1 = \frac{\dot V_1}{A_{c,1}} = \frac{\dot V_1}{\pi D_1^2/4} \quad\rightarrow\quad V_1 = \frac{\dot V_1}{\pi(0.03\text{m})^2/4} \qquad (5)$$

$$V_2 = \frac{\dot V_2}{A_{c,2}} = \frac{\dot V_2}{\pi D_2^2/4} \quad\rightarrow\quad V_2 = \frac{\dot V_2}{\pi(0.05\text{m})^2/4} \qquad (6)$$

$$\text{Re}_1 = \frac{\rho V_1 D_1}{\mu} \quad\rightarrow\quad \text{Re}_1 = \frac{(998\text{ kg/m}^3)V_1(0.03\text{ m})}{1.002\times10^{-3}\text{ kg/m}\cdot s} \qquad (7)$$

$$\text{Re}_2 = \frac{\rho V_2 D_2}{\mu} \quad\rightarrow\quad \text{Re}_2 = \frac{(998\text{ kg/m}^3)V_2(0.05\text{ m})}{1.002\times10^{-3}\text{ kg/m}\cdot s} \qquad (8)$$

$$\frac{1}{\sqrt{f_1}} = -2.0\log\left(\frac{\varepsilon/D_1}{3.7} + \frac{2.51}{\text{Re}_1\sqrt{f_1}}\right) \quad\rightarrow\quad \frac{1}{\sqrt{f_1}} = -2.0\log\left(0 + \frac{2.51}{\text{Re}_1\sqrt{f_1}}\right) \qquad (9)$$

$$\frac{1}{\sqrt{f_2}} = -2.0\log\left(\frac{\varepsilon/D_2}{3.7} + \frac{2.51}{\text{Re}_2\sqrt{f_2}}\right) \quad\rightarrow\quad \frac{1}{\sqrt{f_2}} = -2.0\log\left(0 + \frac{2.51}{\text{Re}_2\sqrt{f_2}}\right) \qquad (10$$

$$h_{L,1} = f_1 \frac{L_1}{D_1}\frac{V_1^2}{2g} \quad\rightarrow\quad h_{L,1} = f_1 \frac{25\text{ m}}{0.03\text{ m}}\frac{V_1^2}{2(9.81\text{ m/s}^2)} \qquad (11)$$

$$h_{L,2} = f_2 \frac{L_2}{D_2}\frac{V_2^2}{2g} \quad\rightarrow\quad h_{L,2} = f_2 \frac{25\text{ m}}{0.05\text{ m}}\frac{V_2^2}{2(9.81\text{ m/s}^2)} \qquad (12)$$

$$\dot{V} = \dot{V}_1 + \dot{V}_2 \qquad (13)$$

This is a system of 13 equations in 13 unknowns, and their simultaneous solution by an equation solver gives

$\dot{V} = \mathbf{0.0183\ m^3/s}$, $\dot{V}_1 = \mathbf{0.0037\ m^3/s}$, $\dot{V}_2 = \mathbf{0.0146\ m^3/s}$,

$V_1$ = 5.30 m/s, $V_2$ = 7.42 m/s, $h_L = h_{L,1} = h_{L,2} = 19.5$ m, $h_{\text{pump,u}}$ = 26.5 m

$\text{Re}_1$ = 158,300, $\text{Re}_2$ = 369,700, $f_1$ = 0.0164, $f_2$ = 0.0139

Note that Re > 4000 for both pipes, and thus the assumption of turbulent flow is verified.

***Discussion*** This problem can also be solved by using an iterative approach, but it will be very time consuming. Equation solvers such as EES are invaluable for this kind of problems.

**8-69E**

**Solution** The flow rate through a piping system connecting two reservoirs is given. The elevation of the source is to be determined.

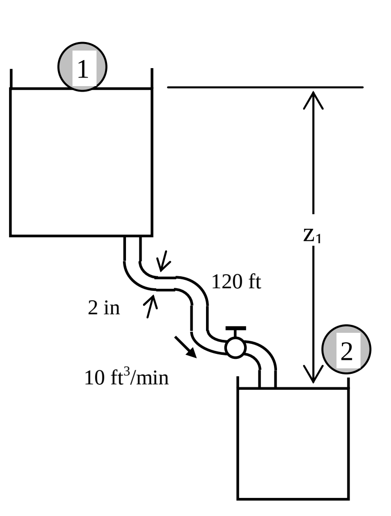
***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The elevations of the reservoirs remain constant. **4** There are no pumps or turbines in the piping system.

***Properties*** The density and dynamic viscosity of water at 70°F are $\rho$ = 62.30 lbm/ft$^3$ and $\mu$ = 2.360 lbm/ft·h = 6.556×10$^{-4}$ lbm/ft·s. The roughness of cast iron pipe is $\varepsilon$ = 0.00085 ft.

***Analysis*** The piping system involves 120 ft of 2-in diameter piping, a well-rounded entrance ($K_L$ = 0.03), 4 standard flanged elbows ($K_L$ = 0.3 each), a fully open gate valve ($K_L$ = 0.2), and a sharp-edged exit ($K_L$ = 1.0). We choose points 1 and 2 at the free surfaces of the two reservoirs. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$), the fluid velocities at both points are zero ($V_1 = V_2 = 0$), the free surface of the lower reservoir is the reference level ($z_2$ = 0), and that there is no pump or turbine ($h_{pump,u} = h_{turbine} = 0$), the energy equation for a control volume between these two points simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{pump,\,u}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{turbine,\,e}+h_L \quad\rightarrow\quad z_1=h_L$$

where $h_L = h_{L,total} = h_{L,major} + h_{L,minor} = \left(f\frac{L}{D}+\sum K_L\right)\frac{V^2}{2g}$




since the diameter of the piping system is constant. The average velocity in the pipe and the Reynolds number are

$$V=\frac{\dot V}{A_c}=\frac{\dot V}{\pi D^2/4}=\frac{10/60\text{ ft}^3/\text{s}}{\pi(2/12\text{ ft})^2/4}=7.64\text{ ft/s}$$

$$\text{Re}=\frac{\rho VD}{\mu}=\frac{(62.3\text{ lbm/ft}^3)(7.64\text{ ft/s})(2/12\text{ ft})}{1.307\times10^{-3}\text{ lbm/ft}\cdot\text{s}}=60{,}700$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D=\frac{0.00085\text{ ft}}{2/12\text{ ft}}=0.0051$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt f}=-2.0\log\left(\frac{\varepsilon/D}{3.7}+\frac{2.51}{\text{Re}\sqrt f}\right)\ \rightarrow\ \frac{1}{\sqrt f}=-2.0\log\left(\frac{0.0051}{3.7}+\frac{2.51}{60{,}700\sqrt f}\right)$$

It gives $f$ = 0.0320. The sum of the loss coefficients is

$$\sum K_L=K_{L,entrance}+4K_{L,elbow}+K_{L,valve}+K_{L,exit}=0.03+4\times0.3+0.2+1.0=2.43$$

Then the total head loss and the elevation of the source become

$$h_L=\left(f\frac{L}{D}+\sum K_L\right)\frac{V^2}{2g}=\left((0.0320)\frac{120\text{ ft}}{2/12\text{ ft}}+2.43\right)\frac{(7.64\text{ ft/s})^2}{2(32.2\text{ ft/s}^2)}=23.1\text{ ft}$$

$$z_1=h_L=\mathbf{23.1\text{ ft}}$$

Therefore, the free surface of the first reservoir must be 23.1 ft above the free surface of the lower reservoir to ensure water flow between the two reservoirs at the specified rate.

***Discussion*** Note that $fL/D$ = 23.0 in this case, which is almost 10 folds of the total minor loss coefficient. Therefore, ignoring the sources of minor losses in this case would result in an error of about 10%.

---

**8-70**

**Solution** A water tank open to the atmosphere is initially filled with water. A sharp-edged orifice at the bottom drains to the atmosphere. The initial velocity from the tank and the time required to empty the tank are to be determined.

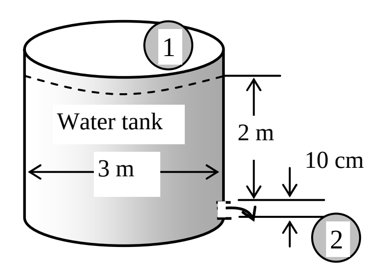
***Assumptions*** **1** The flow is uniform and incompressible. **2** The flow is turbulent so that the tabulated value of the loss coefficient can be used. **3** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Properties*** The loss coefficient is $K_L = 0.5$ for a sharp-edged entrance.

***Analysis*** (*a*) We take point 1 at the free surface of the tank, and point 2 at the exit of the orifice. We also take the reference level at the centerline of the orifice ($z_2 = 0$), and take the positive direction of $z$ to be upwards. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad z_1 = \alpha_2 \frac{V_2^2}{2g} + h_L$$

where the head loss is expressed as $h_L = K_L \dfrac{V^2}{2g}$. Substituting and solving for $V_2$ gives




$$z_1 = \alpha_2 \frac{V_2^2}{2g} + K_L \frac{V_2^2}{2g} \quad \rightarrow \quad 2gz_1 = V_2^2(\alpha_2 + K_L) \quad \rightarrow \quad V_2 = \sqrt{\frac{2gz_1}{\alpha_2 + K_L}}$$

where $\alpha_2 = 1$. Noting that initially $z_1 = 2$ m, the initial velocity is determined to be

$$V_2 = \sqrt{\frac{2gz_1}{1+K_L}} = \sqrt{\frac{2(9.81\text{ m/s}^2)(2\text{ m})}{1+0.5}} = \mathbf{5.11\ m/s}$$

The average discharge velocity through the orifice at any given time, in general, can be expressed as

$$V_2 = \sqrt{\frac{2gz}{1+K_L}}$$

where $z$ is the water height relative to the center of the orifice at that time.

(*b*) We denote the diameter of the orifice by $D$, and the diameter of the tank by $D_0$. The flow rate of water from the tank can be obtained by multiplying the discharge velocity by the orifice area,

$$\dot{V} = A_{\text{orifice}} V_2 = \frac{\pi D^2}{4} \sqrt{\frac{2gz}{1+K_L}}$$

Then the amount of water that flows through the orifice during a differential time interval $dt$ is

$$dV = \dot{V}dt = \frac{\pi D^2}{4} \sqrt{\frac{2gz}{1+K_L}} dt \qquad (1)$$

which, from conservation of mass, must be equal to the decrease in the volume of water in the tank,

$$dV = A_{\text{tank}}(-dz) = -\frac{\pi D_0^2}{4} dz \qquad (2)$$

where $dz$ is the change in the water level in the tank during $dt$. (Note that $dz$ is a negative quantity since the positive direction of $z$ is upwards. Therefore, we used $-dz$ to get a positive quantity for the amount of water discharged). Setting Eqs. (1) and (2) equal to each other and rearranging,

$$\frac{\pi D^2}{4} \sqrt{\frac{2gz}{1+K_L}} dt = -\frac{\pi D_0^2}{4} dz \quad \rightarrow \quad dt = -\frac{D_0^2}{D^2} \sqrt{\frac{1+K_L}{2gz}} dz \quad \rightarrow \quad dt = -\frac{D_0^2}{D^2} \sqrt{\frac{1+K_L}{2g}} z^{-1/2} dz$$

The last relation can be integrated easily since the variables are separated. Letting $t_f$ be the discharge time and integrating it from $t = 0$ when $z = z_1$ to $t = t_f$ when $z = 0$ (completely drained tank) gives

$$\int_{t=0}^{t_f} dt = -\frac{D_0^2}{D^2} \sqrt{\frac{1+K_L}{2g}} \int_{z=z_1}^{0} z^{-1/2} dz \quad \rightarrow \quad t_f = -\frac{D_0^2}{D^2} \sqrt{\frac{1+K_L}{2g}} \left. \frac{z^{-\frac{1}{2}+1}}{-\frac{1}{2}+1} \right|_{z_1}^{0} = \frac{2D_0^2}{D^2} \sqrt{\frac{1+K_L}{2g}} z_1^{1/2}$$

Simplifying and substituting the values given, the draining time is determined to be

$$t_f = \frac{D_0^2}{D^2}\sqrt{\frac{2z_1(1+K_L)}{g}} = \frac{(3\,\text{m})^2}{(0.1\,\text{m})^2}\sqrt{\frac{2(2\,\text{m})(1+0.5)}{9.81\,\text{m/s}^2}} = \mathbf{704\ s} = \mathbf{11.7\ min}$$

***Discussion*** The effect of the loss coefficient $K_L$ on the draining time can be assessed by setting it equal to zero in the draining time relation. It gives

$$t_{f,\text{zero loss}} = \frac{D_0^2}{D^2}\sqrt{\frac{2z_1}{g}} = \frac{(3\,\text{m})^2}{(0.1\,\text{m})^2}\sqrt{\frac{2(2\,\text{m})}{9.81\,\text{m/s}^2}} = 575\,\text{s} = 9.6\,\text{min}$$

Note that the loss coefficient causes the draining time of the tank to increase by (11.7- 9.6)/11.7 = 0.18 or 18%, which is quite **significant**. Therefore, the loss coefficient should always be considered in draining processes.

---

**8-71**

**Solution** A water tank open to the atmosphere is initially filled with water. A sharp-edged orifice at the bottom drains to the atmosphere through a long pipe. The initial velocity from the tank and the time required to empty the tank are to be determined.

***Assumptions*** **1** The flow is uniform and incompressible. **2** The draining pipe is horizontal. **3** The flow is turbulent so that the tabulated value of the loss coefficient can be used. **4** The friction factor remains constant (in reality, it changes since the flow velocity and thus the Reynolds number changes). **5** The effect of the kinetic energy correction factor is negligible, so we set $\alpha = 1$.

***Properties*** The loss coefficient is $K_L = 0.5$ for a sharp-edged entrance. The friction factor of the pipe is given to be 0.015.

***Analysis*** (*a*) We take point 1 at the free surface of the tank, and point 2 at the exit of the pipe. We take the reference level at the centerline of the pipe ($z_2 = 0$), and take the positive direction of $z$ to be upwards. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad z_1=\alpha_2\frac{V_2^2}{2g}+h_L$$

where

$$h_L=h_{L,\text{total}}=h_{L,\text{major}}+h_{L,\text{minor}}=\left(f\frac{L}{D}+\sum K_L\right)\frac{V^2}{2g}=\left(f\frac{L}{D}+K_L\right)\frac{V^2}{2g}$$

since the diameter of the piping system is constant. Substituting and solving for $V_2$ gives

$$z_1=\alpha_2\frac{V_2^2}{2g}+\left(f\frac{L}{D}+K_L\right)\frac{V_2^2}{2g} \quad\rightarrow\quad V_2=\sqrt{\frac{2gz_1}{\alpha_2+fL/D+K_L}}$$

where $\alpha_2 = 1$. Noting that initially $z_1 = 2$ m, the initial velocity is determined to be

$$V_{2,i}=\sqrt{\frac{2gz_1}{1+fL/D+K_L}}=\sqrt{\frac{2(9.81\text{ m/s}^2)(2\text{ m})}{1+0.015(100\text{ m})/(0.1\text{ m})+0.5}}=\mathbf{1.54\ m/s}$$



The average discharge velocity at any given time, in general, can be expressed as

$$V_2=\sqrt{\frac{2gz}{1+fL/D+K_L}}$$

where $z$ is the water height relative to the center of the orifice at that time.

(*b*) We denote the diameter of the pipe by $D$, and the diameter of the tank by $D_o$. The flow rate of water from the tank can be obtained by multiplying the discharge velocity by the pipe cross-sectional area,

$$\dot{V}=A_{pipe}V_2=\frac{\pi D^2}{4}\sqrt{\frac{2gz}{1+fL/D+K_L}}$$

Then the amount of water that flows through the pipe during a differential time interval $dt$ is

$$dV=\dot{V}dt=\frac{\pi D^2}{4}\sqrt{\frac{2gz}{1+fL/D+K_L}}dt \qquad (1)$$

which, from conservation of mass, must be equal to the decrease in the volume of water in the tank,

$$dV=A_{\tan k}(-dz)=-\frac{\pi D_0^2}{4}dz \qquad (2)$$

where $dz$ is the change in the water level in the tank during $dt$. (Note that $dz$ is a negative quantity since the positive direction of $z$ is upwards. Therefore, we used $-dz$ to get a positive quantity for the amount of water discharged). Setting Eqs. (1) and (2) equal to each other and rearranging,

$\frac{\pi D^2}{4}\sqrt{\frac{2gz}{1+fL/D+K_L}}dt=-\frac{\pi D_0^2}{4}dz \rightarrow dt=-\frac{D_0^2}{D^2}\sqrt{\frac{1+fL/D+K_L}{2gz}}dz=-\frac{D_0^2}{D^2}\sqrt{\frac{1+fL/D+K_L}{2g}}z^{-\frac{1}{2}}dz$ The last relation can be integrated easily since the variables are separated. Letting $t_f$ be the discharge time and integrating it from $t = 0$ when $z = z_1$ to $t = t_f$ when $z = 0$ (completely drained tank) gives

$\int_{t=0}^{t_f} dt=-\frac{D_0^2}{D^2}\sqrt{\frac{1+fL/D+K_L}{2g}}\int_{z=z_1}^{0} z^{-1/2}dz \rightarrow t_f=-\frac{D_0^2}{D^2}\sqrt{\frac{1+fL/D+K_L}{2g}}\left.\frac{z^{\frac{1}{2}}}{\frac{1}{2}}\right|_{z_1}^{0}=\frac{2D_0^2}{D^2}\sqrt{\frac{1+fL/D+K_L}{2g}}z_1^{\frac{1}{2}}$ Simplifying and substituting the values given, the draining time is determined to be

$$t_f=\frac{D_0^2}{D^2}\sqrt{\frac{2z_1(1+fL/D+K_L)}{g}}=\frac{(3\,\text{m})^2}{(0.1\,\text{m})^2}\sqrt{\frac{2(2\,\text{m})[1+(0.015)(100\,\text{m})/(0.1\,\text{m})+0.5]}{9.81\,\text{m/s}^2}}=2334\,\text{s}=\mathbf{38.9\ min}$$

***Discussion*** It can be shown by setting $L = 0$ that the draining time without the pipe is only 11.7 min. Therefore, the pipe in this case increases the draining time by more than 3 folds.

**8-72**

**Solution** A water tank open to the atmosphere is initially filled with water. A sharp-edged orifice at the bottom drains to the atmosphere through a long pipe equipped with a pump. For a specified initial velocity, the required useful pumping power and the time required to empty the tank are to be determined.

***Assumptions*** **1** The flow is uniform and incompressible. **2** The draining pipe is horizontal. **3** The flow is turbulent so that the tabulated value of the loss coefficient can be used. **4** The friction factor remains constant. **5** The effect of the kinetic energy correction factor is negligible, so we set $\alpha = 1$.

***Properties*** The loss coefficient is $K_L = 0.5$ for a sharp-edged entrance. The friction factor of the pipe is given to be 0.015. The density of water at 30°C is $\rho = 996$ kg/m$^3$.

***Analysis*** (*a*) We take point 1 at the free surface of the tank, and point 2 at the exit of the pipe. We take the reference level at the centerline of the orifice ($z_2 = 0$), and take the positive direction of $z$ to be upwards. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad z_1+h_{\text{pump, u}}=\alpha_2\frac{V_2^2}{2g}+h_L$$

where $\alpha_2 = 1$ and

$$h_L=h_{L,\text{total}}=h_{L,\text{major}}+h_{L,\text{minor}}=\left(f\frac{L}{D}+\sum K_L\right)\frac{V^2}{2g}=\left(f\frac{L}{D}+K_L\right)\frac{V^2}{2g}$$

since the diameter of the piping system is constant. Substituting and noting that the initial discharge velocity is 4 m/s, the required useful pumping head and power are determined to be

$$\dot m=\rho A_c V_2=\rho(\pi D^2/4)V_2=(996\text{ kg/m}^3)[\pi(0.1\text{ m})^2/4](4\text{ m/s})=31.3\text{ kg/m}^3$$

$$h_{\text{pump, u}}=\left(1+f\frac{L}{D}+K_L\right)\frac{V_2^2}{2g}-z_1=\left(1+(0.015)\frac{100\text{ m}}{0.1\text{ m}}+0.5\right)\frac{(4\text{ m/s})^2}{2(9.81\text{ m/s}^2)}-(2\text{ m})=11.46\text{ m}$$

$$\dot W_{\text{pump, u}}=\dot V\Delta P=\dot m g h_{\text{pump, u}}=(31.3\text{ kg/s})(9.81\text{ m/s}^2)(11.46\text{ m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}}\right)=\mathbf{3.52\ kW}$$

Therefore, the pump must supply 3.52 kW of mechanical energy to water. Note that the shaft power of the pump must be greater than this to account for the pump inefficiency.

(*b*) When the discharge velocity remains constant, the flow rate of water becomes

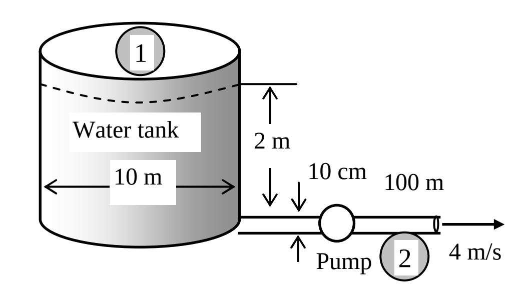
$$\dot V=A_cV_2=(\pi D^2/4)V_2=[\pi(0.1\text{ m})^2/4](4\text{ m/s})=0.03142\text{ m}^3/\text{s}$$

The volume of water in the tank is

$$V=A_{\text{tank}}z_1=(\pi D_0^2/4)z_1=[\pi(3\text{ m})^2/4](2\text{ m})=14.14\text{ m}^3$$

Then the discharge time becomes

$$\Delta t=\frac{V}{\dot V}=\frac{14.14\text{ m}^3}{0.03142\text{ m}^3/\text{s}}=450\text{ s}=\mathbf{7.5\ min}$$




***Discussion***

**1** Note that the pump reduces the discharging time from 38.9 min to 7.5 min. The assumption of constant discharge velocity can be justified on the basis of the pump head being much larger than the elevation head (therefore, the pump will dominate the discharging process). The answer obtained assumes that the elevation head remains constant at 2 m (rather than decreasing to zero eventually), and thus it under predicts the actual discharge time. By an exact analysis, it can be shown that when the effect of the decrease in elevation is considered, the discharge time becomes 468 s = 7.8 min. This is demonstrated below.

**2** The required pump head (of water) is 11.46 m, which is more than 10.3 m of water column which corresponds to the atmospheric pressure at sea level. If the pump exit is at 1 atm, then the absolute pressure at pump inlet must be negative ( = -1.16 m or – 11.4 kPa), which is impossible. Therefore, the system cannot work if the pump is installed near the pipe exit,

and cavitation will occur long before the pipe exit where the pressure drops to 4.2 kPa and thus the pump must be installed close to the pipe entrance. A detailed analysis is given below.

**Demonstration 1 for Prob. 8-72 (extra) (the effect of drop in water level on discharge time)**

Noting that the water height *z* in the tank is variable, the average discharge velocity through the pipe at any given time, in general, can be expressed as

$$h_{\text{pump, u}} = \left(1 + f\frac{L}{D} + K_L\right)\frac{V_2^2}{2g} - z \quad \rightarrow \quad V_2 = \sqrt{\frac{2g(z + h_{\text{pump, u}})}{1 + fL/D + K_L}}$$

where *z* is the water height relative to the center of the orifice at that time. We denote the diameter of the pipe by *D*, and the diameter of the tank by $D_0$. The flow rate of water from the tank can be obtained by multiplying the discharge velocity by the cross-sectional area of the pipe,

$$\dot{V} = A_{pipe}V_2 = \frac{\pi D^2}{4}\sqrt{\frac{2g(z + h_{\text{pump, u}})}{1 + fL/D + K_L}}$$

Then the amount of water that flows through the orifice during a differential time interval *dt* is

$$dV = \dot{V}dt = \frac{\pi D^2}{4}\sqrt{\frac{2g(z + h_{\text{pump, u}})}{1 + fL/D + K_L}}dt \qquad (1)$$

which, from conservation of mass, must be equal to the decrease in the volume of water in the tank,

$$dV = A_{\tan k}(-dz) = -\frac{\pi D_0^2}{4}dz \qquad (2)$$

where *dz* is the change in the water level in the tank during *dt*. (Note that *dz* is a negative quantity since the positive direction of *z* is upwards. Therefore, we used *–dz* to get a positive quantity for the amount of water discharged). Setting Eqs. (1) and (2) equal to each other and rearranging,

$$\frac{\pi D^2}{4}\sqrt{\frac{2g(z + h_{\text{pump,u}})}{1 + fL/D + K_L}}dt = -\frac{\pi D_0^2}{4}dz \quad \rightarrow dt = -\frac{D_0^2}{D^2}\sqrt{\frac{1 + fL/D + K_L}{2g}}(z + h_{\text{pump,u}})^{-\frac{1}{2}}dz$$

The last relation can be integrated easily since the variables are separated. Letting $t_f$ be the discharge time and integrating it from $t = 0$ when $z = z_1$ to $t = t_f$ when $z = 0$ (completely drained tank) gives

$$\int_{t=0}^{t_f} dt = -\frac{D_0^2}{D^2}\sqrt{\frac{1 + fL/D + K_L}{2g}}\int_{z=z_1}^{0}(z + h_{\text{pump, u}})^{-1/2}dz$$

Performing the integration gives

$$t_f = -\frac{D_0^2}{D^2}\sqrt{\frac{1 + fL/D + K_L}{2g}}\left.\frac{(z + h_{\text{pump}})^{\frac{1}{2}}}{\frac{1}{2}}\right|_{z_1}^{0} = \frac{D_0^2}{D^2}\left(\sqrt{\frac{2(z_1 + h_{\text{pump}})(1 + fL/D + K_L)}{g}} - \sqrt{\frac{2h_{\text{pump}}(1 + fL/D + K_L)}{g}}\right)$$

Substituting the values given, the draining time is determined to be

$$t_f = \frac{(3\,\text{m})^2}{(0.1\,\text{m})^2}\left(\sqrt{\frac{2(2 + 11.46\text{ m})[1 + 0.015 \times 100/0.1 + 0.5]}{9.81\,\text{m/s}^2}} - \sqrt{\frac{2(11.46\text{ m})[1 + 0.015 \times 100/0.1 + 0.5]}{9.81\,\text{m/s}^2}}\right)$$

$$= 468\,\text{s} = \mathbf{7.8\ min}$$

**Demonstration 2 for Prob. 8-72 (on cavitation)**

We take the pump as the control volume, with point 1 at the inlet and point 2 at the exit. We assume the pump inlet and outlet diameters to be the same and the elevation difference between the pump inlet and the exit to be negligible. Then we have $z_1 \cong z_2$ and $V_1 \cong V_2$. The pump is located near the pipe exit, and thus the pump exit pressure is equal to the pressure at the pipe exit, which is the atmospheric pressure, $P_2 = P_{\text{atm}}$. Also, the can take $h_L = 0$ since the frictional effects and loses in the pump are accounted for by the pump efficiency. Then the energy equation for the pump (in terms of heads) reduces to

$$\frac{P_1}{\rho g} + \alpha_1\frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2\frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad \frac{P_{1,\text{abs}}}{\rho g} + h_{\text{pump, u}} = \frac{P_{\text{atm}}}{\rho g}$$

Solving for $P_1$ and substituting,

$$P_{1,\text{abs}} = P_{\text{atm}} - \rho g h_{\text{pump, u}}$$

$$= (101.3\ \text{kPa}) - (996\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(11.46\ \text{m})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1\ \text{kN/m}^2}\right) = \mathbf{-10.7\ kPa}$$

which is impossible (absolute pressure cannot be negative). The technical answer to the question is that cavitation **will occur** since the pressure drops below the vapor pressure of 4.246 kPa. The practical answer is that the question is invalid (void) since the system will not work anyway. Therefore, we conclude that the pump must be located **near the beginning**, not the end of the pipe. Note that when doing a cavitation analysis, we must work with the absolute pressures. (If the system were installed as indicated, a water velocity of $V$ = 4 m/s could not be established regardless of how much pump power were applied. This is because the atmospheric air and water elevation heads alone are not sufficient to drive such flow, with the pump restoring pressure after the flow.)

To determine the furthest distance from the tank the pump can be located without allowing cavitation, we assume the pump is located at a distance $L^*$ from the exit, and choose the pump and the discharge portion of the pipe (from the pump to the exit) as the system, and write the energy equation. The energy equation this time will be as above, except that $h_L$ (the pipe losses) must be considered and the pressure at 1 (pipe inlet) is the cavitation pressure, $P_1$ = 4.246 kPa:

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad \frac{P_{1,\text{abs}}}{\rho g} + h_{\text{pump, u}} = \frac{P_{\text{atm}}}{\rho g} + f\frac{L^*}{D}\frac{V^2}{2g}$$

or $$f\frac{L^*}{D}\frac{V^2}{2g} = \frac{P_{1,\text{abs}} - P_{\text{atm}}}{\rho g} + h_{\text{pump, u}}$$

Substituting the given values and solving for $L^*$ gives

$$(0.015)\frac{L^*}{0.1\ \text{m}}\frac{(4\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)} = \frac{(4.246 - 101.3)\ \text{kN/m}^2}{(996\ \text{kg/m}^3)(9.81\ \text{m/s}^2)}\left(\frac{1000\ \text{kg}\cdot\text{m/s}^2}{1\ \text{kN}}\right) + (11.46\ \text{m}) \quad \rightarrow \quad L^* = 12.5\ \text{m}$$

Therefore, the pump must be at least 12.5 m from the pipe exit to avoid cavitation at the pump inlet (this is where the lowest pressure occurs in the piping system, and where the cavitation is most likely to occur).

Cavitation onset places an upper limit to the length of the pipe on the suction side. A pipe slightly longer would become vapor bound, and the pump could not pull the suction necessary to sustain the flow. Even if the pipe on the suction side were slightly shorter than 100 – 12.5 = 87.5 m, cavitation can still occur in the pump since the liquid in the pump is usually accelerated at the expense of pressure, and cavitation in the pump could erode and destroy the pump.

Also, over time, scale and other buildup inside the pipe can and will increase the pipe roughness, increasing the friction factor $f$, and therefore the losses. Buildup also decreases the pipe diameter, which increases pressure drop. Therefore, flow conditions and system performance may change (generally decrease) as the system ages. A new system that marginally misses cavitation may degrade to where cavitation becomes a problem. Proper design avoids these problems, or where cavitation cannot be avoided for some reason, it can at least be anticipated.

**8-73**

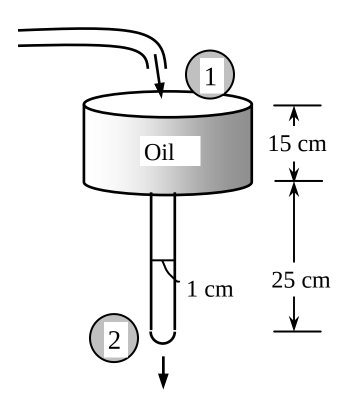
**Solution** Oil is flowing through a vertical glass funnel which is always maintained full. The flow rate of oil through the funnel and the funnel effectiveness are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed (to be verified). **3** The frictional loses in the cylindrical reservoir are negligible since its diameter is very large and thus the oil velocity is very low.

***Properties*** The density and viscosity of oil at 20°C are $\rho$ = 888.1 kg/m$^3$ and $\mu$ = 0.8374 kg/m·s.

***Analysis*** We take point 1 at the free surface of the oil in the cylindrical reservoir, and point 2 at the exit of the funnel pipe which is also taken as the reference level ($z_2$ = 0). The fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is negligible ($V_1 \cong 0$). For the ideal case of "frictionless flow," the exit velocity is determined from the Bernoulli equation to be

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad \rightarrow \quad V_2=V_{2,\max}=\sqrt{2gz_1}$$




Substituting,

$$V_{2,\max}=\sqrt{2gz_1}=\sqrt{2(9.81\text{ m/s}^2)(0.40\text{ m})}=2.801\text{ m/s}$$

This is the flow velocity for the frictionless case, and thus it is the maximum flow velocity. Then the maximum flow rate and the Reynolds number become

$$\dot{V}_{\max}=V_{2,\max}A_2=V_{2,\max}(\pi D_2^2/4)$$
$$=(2.801\text{ m/s})[\pi(0.01\text{ m})^2/4]=2.20\times10^{-4}\text{ m}^3\text{/s}$$

$$\text{Re}=\frac{\rho VD}{\mu}=\frac{(888.1\text{ kg/m}^3)(2.801\text{ m/s})(0.01\text{ m})}{0.8374\text{ kg/m}\cdot\text{s}}=29.71$$

which is less than 2300. Therefore, the flow is laminar, as postulated. (Note that in the actual case the velocity and thus the Reynolds number will be even smaller, verifying the flow is always laminar). The entry length in this case is

$$L_h=0.05\,\text{Re}\,D=0.05\times29.71\times(0.01\text{ m})=0.015\text{ m}$$

which is much less than the 0.25 m pipe length. Therefore, the entrance effects can be neglected as postulated.

Noting that the flow through the pipe is laminar and can be assumed to be fully developed, the flow rate can be determined from the appropriate relation with θ = -90° since the flow is downwards in the vertical direction,

$$\dot{V}=\frac{(\Delta P-\rho gL\sin\theta)\pi D^4}{128\mu L}$$

where $\Delta P=P_{\text{pipe inlet}}-P_{\text{pipe exit}}=(P_{atm}+\rho gh_{\text{cylinder}})-P_{atm}=\rho gh_{\text{cylinder}}$ is the pressure difference across the pipe, $L = h_{\text{pipe}}$, and sin θ = sin (-90°) = -1. Substituting, the flow rate is determined to be

$$\dot{V}=\frac{\rho g(h_{\text{cylinder}}+h_{\text{pipe}})\pi D^4}{128\mu L}=\frac{(888.1\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.15+0.25\text{ m})\pi(0.01\text{ m})^4}{128(0.8374\text{ kg/m}\cdot\text{s})(0.25\text{ m})}=\mathbf{4.09\times10^{-6}\ m^3/s}$$

Then the "funnel effectiveness" becomes

$$\text{Eff}=\frac{\dot{V}}{\dot{V}_{\max}}=\frac{4.09\times10^{-6}\text{ m}^3\text{/s}}{2.20\times10^{-4}\text{ m}^3\text{/s}}=0.0186 \quad \text{or} \quad \mathbf{1.86\%}$$

***Discussion*** Note that the flow is driven by gravity alone, and the actual flow rate is a small fraction of the flow rate that would have occurred if the flow were frictionless.

**8-74**

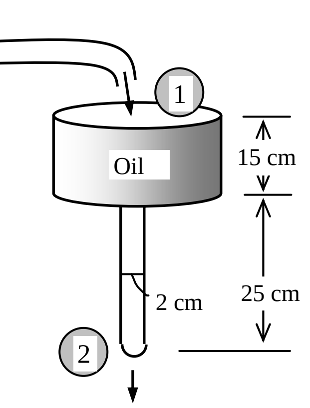
**Solution** Oil is flowing through a vertical glass funnel which is always maintained full. The flow rate of oil through the funnel and the funnel effectiveness are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed (to be verified). **3** The frictional loses in the cylindrical reservoir are negligible since its diameter is very large and thus the oil velocity is very low.

***Properties*** The density and viscosity of oil at 20°C are $\rho$ = 888.1 kg/m$^3$ and $\mu$ = 0.8374 kg/m·s.

***Analysis*** We take point 1 at the free surface of the oil in the cylindrical reservoir, and point 2 at the exit of the funnel pipe, which is also taken as the reference level ($z_2$ = 0). The fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is negligible ($V_1 \cong 0$). For the ideal case of "frictionless flow," the exit velocity is determined from the Bernoulli equation to be

$$\frac{P_1}{\rho g}+\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\frac{V_2^2}{2g}+z_2 \quad\rightarrow\quad V_2=V_{2,\max}=\sqrt{2gz_1}$$




(*a*) **Case 1**: *Pipe length remains constant at 25 cm, but the pipe diameter is doubled* to $D_2$ = 2 cm:

Substitution gives

$$V_{2,\max}=\sqrt{2gz_1}=\sqrt{2(9.81\text{ m/s}^2)(0.40\text{ m})}=2.801\text{ m/s}$$

This is the flow velocity for the frictionless case, and thus it is the maximum flow velocity. Then the maximum flow rate and the Reynolds number become

$$\dot{V}_{\max}=V_{2,\max}A_2=V_{2,\max}(\pi D_2^2/4)=(2.801\text{ m/s})[\pi(0.02\text{ m})^2/4]$$
$$=8.80\times10^{-4}\text{ m}^3\text{/s}$$

$$\text{Re}=\frac{\rho VD}{\mu}=\frac{(888.1\text{ kg/m}^3)(2.801\text{ m/s})(0.02\text{ m})}{0.8374\text{ kg/m}\cdot\text{s}}=59.41$$

which is less than 2300. Therefore, the flow is laminar. (Note that in the actual case the velocity and thus the Reynolds number will be even smaller, verifying the flow is always laminar). The entry length is

$$L_h=0.05\,\text{Re}\,D=0.05\times59.41\times(0.02\text{ m})=0.059\text{ m}$$

which is considerably less than the 0.25 m pipe length. Therefore, the entrance effects can be neglected (with reservation).

Noting that the flow through the pipe is laminar and can be assumed to be fully developed, the flow rate can be determined from the appropriate relation with θ = -90° since the flow is downwards in the vertical direction,

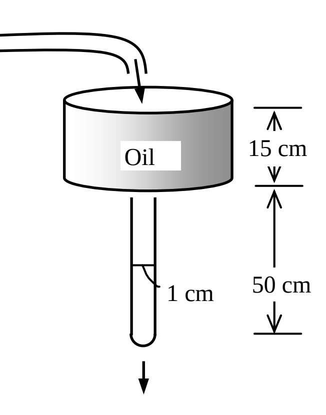
$$\dot{V}=\frac{(\Delta P-\rho gL\sin\theta)\pi D^4}{128\mu L}$$

where $\Delta P=P_{\text{pipe inlet}}-P_{\text{pipe exit}}=(P_{atm}+\rho gh_{cylinder})-P_{atm}=\rho gh_{cylinder}$ is the pressure difference across the pipe, $L = h_{pipe}$, and sin θ = sin (-90°) = -1. Substituting, the flow rate is determined to be

$$\dot{V}=\frac{\rho g(h_{cylinder}+h_{pipe})\pi D^4}{128\mu L}=\frac{(888.1\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.15+0.25\text{ m})\pi(0.02\text{ m})^4}{128(0.8374\text{ kg/m}\cdot\text{s})(0.25\text{ m})}=\mathbf{6.54\times10^{-5}\ m^3/s}$$

Then the "funnel effectiveness" becomes

$$\text{Eff}=\frac{\dot{V}}{\dot{V}_{\max}}=\frac{0.654\times10^{-4}\text{ m}^3\text{/s}}{8.80\times10^{-4}\text{ m}^3\text{/s}}=0.074\quad\text{or}\quad\mathbf{7.4\%}$$




(*b*) **Case 2**: *Pipe diameter remains constant at 1 cm, but the pipe length is doubled to L* = 50 *cm*:

Substitution gives

$$V_{2,\max}=\sqrt{2gz_1}=\sqrt{2(9.81\text{ m/s}^2)(0.65\text{ m})}=3.571\text{ m/s}$$

This is the flow velocity for the frictionless case, and thus it is the maximum flow velocity. Then the maximum flow rate and the Reynolds number become

$$\dot{V}_{max} = V_{2,max} A_2 = V_{2,max} (\pi D_2^2 / 4) = (3.571\,\text{m/s})[\pi (0.01\,\text{m})^2 / 4]$$
$$= 2.805 \times 10^{-4}\ \text{m}^3 / s$$

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(888.1\,\text{kg/m}^3)(3.571\,\text{m/s})(0.01\,\text{m})}{0.8374\,\text{kg/m}\cdot\text{s}} = 37.87$$

which is less than 2300. Therefore, the flow is laminar. (Note that in the actual case the velocity and thus the Reynolds number will be even smaller, verifying the flow is always laminar). The entry length is

$$L_h = 0.05\,\text{Re}\,D = 0.05 \times 37.87 \times (0.01\,\text{m}) = 0.019\,\text{m}$$

which is much less than the 0.50 m pipe length. Therefore, the entrance effects can be neglected.

Noting that the flow through the pipe is laminar and can be assumed to be fully developed, the flow rate can be determined from the appropriate relation with θ = -90° since the flow is downwards in the vertical direction,

$$\dot{V} = \frac{(\Delta P - \rho g L \sin\theta)\pi D^4}{128 \mu L}$$

where $\Delta P = P_{\text{pipe inlet}} - P_{\text{pipe exit}} = (P_{\text{atm}} + \rho g h_{\text{cylinder}}) - P_{\text{atm}} = \rho g h_{\text{cylinder}}$ is the pressure difference across the pipe, $L = h_{\text{pipe}}$, and sin θ = sin (-90°) = -1. Substituting, the flow rate is determined to be

$$\dot{V} = \frac{\rho g (h_{\text{cylinder}} + h_{\text{pipe}})\pi D^4}{128 \mu L} = \frac{(888.1\,\text{kg/m}^3)(9.81\,\text{m/s}^2)(0.15 + 0.50\,\text{m})\pi (0.01\,\text{m})^4}{128(0.8374\,\text{kg/m}\cdot\text{s})(0.50\,\text{m})} = \mathbf{3.32 \times 10^{-6}\ m^3/s}$$

Then the "funnel effectiveness" becomes

$$\text{Eff} = \frac{\dot{V}}{\dot{V}_{max}} = \frac{3.32 \times 10^{-6}\ \text{m}^3/\text{s}}{2.805 \times 10^{-4}\ \text{m}^3/\text{s}} = 0.0118 \quad \text{or} \quad \mathbf{1.18\%}$$

***Discussion*** Note that the funnel effectiveness increases as the pipe diameter is increased, and decreases as the pipe length is increased. This is because the frictional losses are proportional to the length but inversely proportional to the diameter of the flow sections.

**8-75**

**Solution** Water is drained from a large reservoir through two pipes connected in series. The discharge rate of water from the reservoir is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The pipes are horizontal. **3** The entrance effects are negligible, and thus the flow is fully developed. **4** The flow is turbulent so that the tabulated value of the loss coefficients can be used. **5** The pipes involve no components such as bends, valves, and other connectors. **6** The piping section involves no work devices such as pumps and turbines. **7** The reservoir is open to the atmosphere so that the pressure is atmospheric pressure at the free surface. **8** The water level in the reservoir remains constant. **9** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Properties*** The density and dynamic viscosity of water at 15°C are $\rho$ = 999.1 kg/m$^3$ and $\mu$ = 1.138×10$^{-3}$ kg/m·s, respectively. The loss coefficient is $K_L$ = 0.5 for a sharp-edged entrance, and it is 0.46 for the sudden contraction, corresponding to $d^2/D^2 = 4^2/10^2 = 0.16$. The pipes are made of plastic and thus they are smooth, $\varepsilon = 0$.

***Analysis*** We take point 1 at the free surface of the reservoir, and point 2 at the exit of the pipe, which is also taken to be the reference level ($z_2 = 0$). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$), the fluid level in the reservoir is constant ($V_1 = 0$), and that there are no work devices such as pumps and turbines, the energy equation for a control volume between these two points (in terms of heads) simplifies to

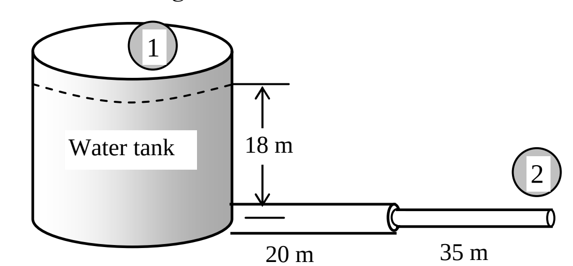
$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad z_1=\alpha_2\frac{V_2^2}{2g}+h_L$$

where $\alpha_2 = 1$. Substituting,

$$18\text{ m}=\frac{V_2^2}{2(9.81\text{m/s}^2)}+h_L \qquad (1)$$

where

$$h_L=h_{L,\text{total}}=h_{L,\text{major}}+h_{L,\text{minor}}=\sum\left(f\frac{L}{D}+\sum K_L\right)\frac{V^2}{2g}$$




Note that the diameters of the two pipes, and thus the flow velocities through them are different. Denoting the first pipe by 1 and the second pipe by 2, and using conservation of mass, the velocity in the first pipe can be expressed in terms of $V_2$ as

$$\dot m_1=\dot m_2 \;\rightarrow\; \rho V_1A_1=\rho V_2A_2 \;\rightarrow\; V_1=\frac{A_2}{A_1}V_2=\frac{D_2^2}{D_1^2}V_2=\frac{(4\text{ cm})^2}{(10\text{ cm})^2}V_2 \;\rightarrow\; V_1=0.16V_2 \qquad (2)$$

Then the head loss can be expressed as

$$h_L=\left(f_1\frac{L_1}{D_1}+K_{L,\text{entrance}}\right)\frac{V_1^2}{2g}+\left(f_2\frac{L_2}{D_2}+K_{L,\text{contraction}}\right)\frac{V_2^2}{2g}$$

or

$$h_L=\left(f_1\frac{20\text{ m}}{0.10\text{ m}}+0.5\right)\frac{V_1^2}{2(9.81\text{m/s}^2)}+\left(f_2\frac{35\text{ m}}{0.04\text{ m}}+0.46\right)\frac{V_2^2}{2(9.81\text{m/s}^2)} \qquad (3)$$

The flow rate, the Reynolds number, and the friction factor are expressed as

$$\dot V=V_2A_2=V_2(\pi D_2^2/4) \quad\rightarrow\quad \dot V=V_2[\pi(0.04\text{ m})^2/4] \qquad (4)$$

$$\text{Re}_1=\frac{\rho V_1D_1}{\mu} \;\rightarrow\; \text{Re}_1=\frac{(999.1\text{ kg/m}^3)\mathbf{V}_1(0.10\text{ m})}{1.138\times10^{-3}\text{ kg/m}\cdot s} \qquad (5)$$

$$\text{Re}_2=\frac{\rho V_2D_2}{\mu} \;\rightarrow\; \text{Re}_2=\frac{(999.1\text{ kg/m}^3)\mathbf{V}_2(0.04\text{ m})}{1.138\times10^{-3}\text{ kg/m}\cdot s} \qquad (6)$$

$$\frac{1}{\sqrt{f_1}}=-2.0\log\left(\frac{\varepsilon/D_1}{3.7}+\frac{2.51}{\text{Re}_1\sqrt{f_1}}\right) \;\rightarrow\; \frac{1}{\sqrt{f_1}}=-2.0\log\left(0+\frac{2.51}{\text{Re}_1\sqrt{f_1}}\right) \qquad (7)$$

$$\frac{1}{\sqrt{f_2}}=-2.0\log\left(\frac{\varepsilon/D_2}{3.7}+\frac{2.51}{\text{Re}_2\sqrt{f_2}}\right) \;\rightarrow\; \frac{1}{\sqrt{f_2}}=-2.0\log\left(0+\frac{2.51}{\text{Re}_2\sqrt{f_2}}\right) \qquad (8)$$

This is a system of 8 equations in 8 unknowns, and their simultaneous solution by an equation solver gives

$\dot{V} = \mathbf{0.00595\ m^3/s}$, $V_1$ = 0.757 m/s, $V_2$ = 4.73 m/s, $h_L = h_{L1} + h_{L2}$ = 0.13 + 16.73 = 16.86 m,

$Re_1$ = 66,500, $Re_2$ = 166,200, $f_1$ = 0.0196, $f_2$ = 0.0162

Note that Re > 4000 for both pipes, and thus the assumption of turbulent flow is valid.

***Discussion*** This problem can also be solved by using an iterative approach by assuming an exit velocity, but it will be very time consuming. Equation solvers such as EES are invaluable for this kind of problems.

**8-76E**

**Solution** The flow rate through a piping system between a river and a storage tank is given. The power input to the pump is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The flow is turbulent so that the tabulated value of the loss coefficients can be used (to be verified). **4** The elevation difference between the free surfaces of the tank and the river remains constant. **5** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

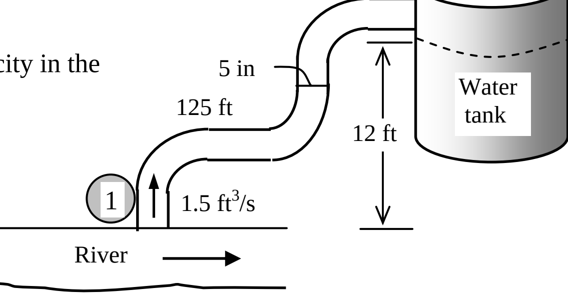
***Properties*** The density and dynamic viscosity of water at 70°F are $\rho$ = 62.30 lbm/ft$^3$ and $\mu$ = 2.360 lbm/ft·h = $6.556\times10^{-4}$ lbm/ft·s. The roughness of galvanized iron pipe is $\varepsilon$ = 0.0005 ft.

***Analysis*** The piping system involves 125 ft of 5-in diameter piping, an entrance with negligible loses, 3 standard flanged 90° smooth elbows ($K_L$ = 0.3 each), and a sharp-edged exit ($K_L$ = 1.0). We choose points 1 and 2 at the free surfaces of the river and the tank, respectively. We note that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$), and the fluid velocity is 6 ft/s at point 1 and zero at point 2 ($V_1$ = 6 ft/s and $V_2$ =0). We take the free surface of the river as the reference level ($z_1$ = 0). Then the energy equation for a control volume between these two points simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad \alpha_1\frac{V_1^2}{2g}+h_{\text{pump, u}}=z_2+h_L$$

where $\alpha_1$ = 1 and

$$h_L=h_{L,\text{total}}=h_{L,\text{major}}+h_{L,\text{minor}}=\left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}$$




since the diameter of the piping system is constant. The average velocity in the pipe and the Reynolds number are

$$V=\frac{\dot V}{A_c}=\frac{\dot V}{\pi D^2/4}=\frac{1.5\text{ ft}^3/\text{s}}{\pi(5/12\text{ ft})^2/4}=11.0\text{ ft/s}$$

$$\text{Re}=\frac{\rho VD}{\mu}=\frac{(62.3\text{ lbm/ft}^3)(11.0\text{ ft/s})(5/12\text{ ft})}{6.556\times10^{-4}\text{ lbm/ft}\cdot\text{s}}=435{,}500$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D=\frac{0.0005\text{ ft}}{5/12\text{ ft}}=0.0012$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt f}=-2.0\log\left(\frac{\varepsilon/D}{3.7}+\frac{2.51}{\text{Re}\sqrt f}\right)\quad\rightarrow\quad\frac{1}{\sqrt f}=-2.0\log\left(\frac{0.0012}{3.7}+\frac{2.51}{435{,}500\sqrt f}\right)$$

It gives $f$ = 0.0211. The sum of the loss coefficients is

$$\sum K_L=K_{L,\text{entrance}}+3K_{L,\text{elbow}}+K_{L,\text{exit}}=0+3\times0.3+1.0=1.9$$

Then the total head loss becomes

$$h_L=\left(f\frac{L}{D}+\sum K_L\right)\frac{V^2}{2g}=\left((0.0211)\frac{125\text{ ft}}{5/12\text{ ft}}+1.90\right)\frac{(11.0\text{ ft/s})^2}{2(32.2\text{ ft/s}^2)}=15.5\text{ ft}$$

The useful pump head input and the required power input to the pump are

$$h_{\text{pump, u}}=z_2+h_L-\frac{V_1^2}{2g}=12+15.5-\frac{(6\text{ ft/s})^2}{2(32.2\text{ ft/s}^2)}=26.9\text{ ft}$$

$$\dot W_{\text{pump}}=\frac{\dot W_{\text{pump, u}}}{\eta_{\text{pump}}}=\frac{\dot V\rho g h_{\text{pump, u}}}{\eta_{\text{pump}}}$$

$$=\frac{(1.5\text{ ft}^3/\text{s})(62.30\text{ lbm/ft}^3)(32.2\text{ ft/s}^2)(26.9\text{ ft})}{0.70}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1\text{ kW}}{737\text{ lbf}\cdot\text{ft/s}}\right)=\mathbf{4.87\text{ kW}}$$

Therefore, 4.87 kW of electric power must be supplied to the pump.

***Discussion*** The friction factor could also be determined easily from the explicit Haaland relation. It would give $f$ = 0.0211, which is identical to the calculated value. The friction coefficient would drop to 0.0135 if smooth pipes were used. Note that $fL/D$ = 6.3 in this case, which is about 3 times the total minor loss coefficient of 1.9. Therefore, the frictional losses in the pipe dominate the minor losses, but the minor losses are still significant.

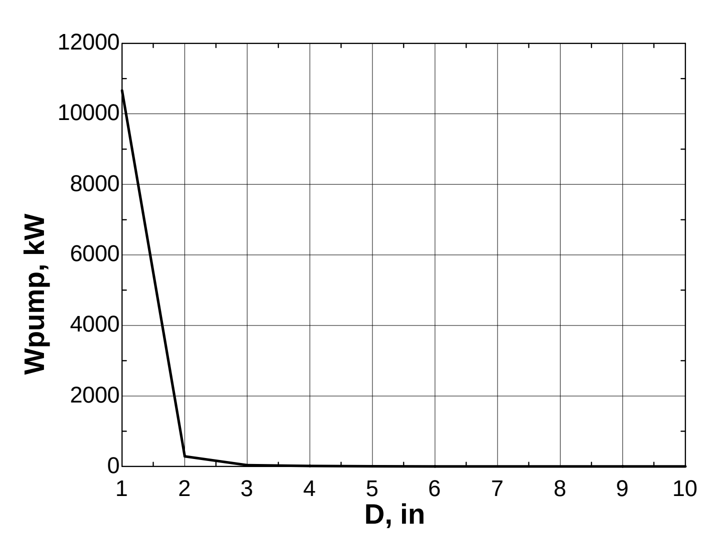
---

**8-77** 

**Solution** In the previous problem, the effect of the pipe diameter on pumping power for the same constant flow rate is to be investigated by varying the pipe diameter from 1 in to 10 in in increments of 1 in.

***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
g=32.2
L= 125
D=Dinch/12
z2= 12
rho=62.30
nu=mu/rho
mu=0.0006556
eff=0.70
Re=V2*D/nu
A=pi*(D^2)/4
V2=Vdot/A
Vdot= 1.5
V1=6
eps1=0.0005
rf1=eps1/D
1/sqrt(f1)=-2*log10(rf1/3.7+2.51/(Re*sqrt(f1)))
KL= 1.9
HL=(f1*(L/D)+KL)*(V2^2/(2*g))
hpump=z2+HL-V1^2/(2*32.2)
Wpump=(Vdot*rho*hpump)/eff/737
```




| $D$, in | $W_{pump}$, kW | $V$, ft/s | Re |
|---|---|---|---|
| 1 | 2.178E+06 | 275.02 | 10667.48 |
| 2 | 1.089E+06 | 68.75 | 289.54 |
| 3 | 7.260E+05 | 30.56 | 38.15 |
| 4 | 5.445E+05 | 17.19 | 10.55 |
| 5 | 4.356E+05 | 11.00 | 4.88 |
| 6 | 3.630E+05 | 7.64 | 3.22 |
| 7 | 3.111E+05 | 5.61 | 2.62 |
| 8 | 2.722E+05 | 4.30 | 2.36 |
| 9 | 2.420E+05 | 3.40 | 2.24 |
| 10 | 2.178E+05 | 2.75 | 2.17 |

***Discussion*** We see that the required pump power decreases very rapidly as pipe diameter increases. This is due to the significant decrease in irreversible head loss in larger diameter pipes.

---

**8-78**

**Solution** A solar heated water tank is to be used for showers using gravity driven flow. For a specified flow rate, the elevation of the water level in the tank relative to showerhead is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The flow is turbulent so that the tabulated value of the loss coefficients can be used (to be verified). **4** The elevation difference between the free surface of water in the tank and the shower head remains constant. **5** There are no pumps or turbines in the piping system. **6** The losses at the entrance and at the showerhead are said to be negligible. **7** The water tank is open to the atmosphere. **8** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Properties*** The density and dynamic viscosity of water at 40°C are $\rho$ = 992.1 kg/m$^3$ and $\mu = 0.653\times10^{-3}$ kg/m·s, respectively. The loss coefficient is $K_L$ = 0.5 for a sharp-edged entrance. The roughness of galvanized iron pipe is $\varepsilon$ = 0.00015 m.

***Analysis*** The piping system involves 20 m of 1.5-cm diameter piping, an entrance with negligible loss, 4 miter bends (90°) without vanes ($K_L$ = 1.1 each), and a wide open globe valve ($K_L$ = 10). We choose point 1 at the free surface of water in the tank, and point 2 at the shower exit, which is also taken to be the reference level ($z_2$ = 0). The fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$), and $V_1$ = 0. Then the energy equation for a control volume between these two points simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad z_1=\alpha_2\frac{V_2^2}{2g}+h_L$$

where $h_L = h_{L,\text{total}} = h_{L,\text{major}} + h_{L,\text{minor}} = \left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}$

since the diameter of the piping system is constant. The average velocity in the pipe and the Reynolds number are

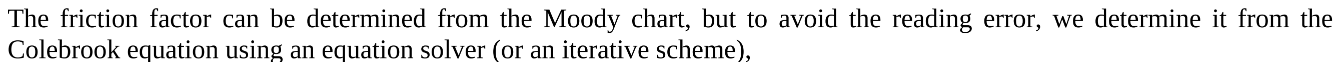
$$V_2=\frac{\dot V}{A_c}=\frac{\dot V}{\pi D^2/4}=\frac{0.0007\ \text{m}^3/\text{s}}{\pi(0.015\ \text{m})^2/4}=3.961\ \text{m/s}$$

$$\text{Re}=\frac{\rho V_2 D}{\mu}=\frac{(992.1\ \text{kg/m}^3)(3.961\ \text{m/s})(0.015\ \text{m})}{0.653\times10^{-3}\ \text{kg/m}\cdot\text{s}}=90{,}270$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D=\frac{0.00015\ \text{m}}{0.015\ \text{m}}=0.01$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),



$$\frac{1}{\sqrt f}=-2.0\log\left(\frac{\varepsilon/D}{3.7}+\frac{2.51}{\text{Re}\sqrt f}\right)\quad\rightarrow\quad\frac{1}{\sqrt f}=-2.0\log\left(\frac{0.0051}{3.7}+\frac{2.51}{90{,}270\sqrt f}\right)$$

It gives $f$ = 0.03857. The sum of the loss coefficients is

$$\sum K_L=K_{L,\text{entrance}}+4K_{L,\text{elbow}}+K_{L,\text{valve}}+K_{L,\text{exit}}=0+4\times1.1+10+0=14.4$$

Note that we do not consider the exit loss unless the exit velocity is dissipated within the system considered (in this case it is not). Then the total head loss and the elevation of the source become

$$h_L=\left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}=\left((0.03857)\frac{20\ \text{m}}{0.015\ \text{m}}+14.4\right)\frac{(3.961\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)}=52.6\ \text{m}$$

$$z_1=\alpha_2\frac{V_2^2}{2g}+h_L=(1)\frac{(3.961\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)}+52.6\ \text{m}=\mathbf{53.4\ m}$$

since $\alpha_2$ = 1. Therefore, the free surface of the tank must be 53.4 m above the shower exit to ensure water flow at the specified rate.

***Discussion*** We neglected the minor loss associated with the shower head. In reality, this loss is most likely significant.

**8-79**

**Solution** The flow rate through a piping system connecting two water reservoirs with the same water level is given. The absolute pressure in the pressurized reservoir is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The flow is turbulent so that the tabulated value of the loss coefficients can be used (to be verified). **4** There are no pumps or turbines in the piping system.

***Properties*** The density and dynamic viscosity of water at 10°C are $\rho$ = 999.7 kg/m$^3$ and $\mu$ = 1.307×10$^{-3}$ kg/m·s. The loss coefficient is $K_L$ = 0.5 for a sharp-edged entrance, $K_L$ = 2 for swing check valve, $K_L$ = 0.2 for the fully open gate valve, and $K_L$ = 1 for the exit. The roughness of cast iron pipe is $\varepsilon$ = 0.00026 m.

***Analysis*** We choose points 1 and 2 at the free surfaces of the two reservoirs. We note that the fluid velocities at both points are zero ($V_1 = V_2$ =0), the fluid at point 2 is open to the atmosphere (and thus $P_2 = P_{atm}$), both points are at the same level ($z_1 = z_2$). Then the energy equation for a control volume between these two points simplifies to

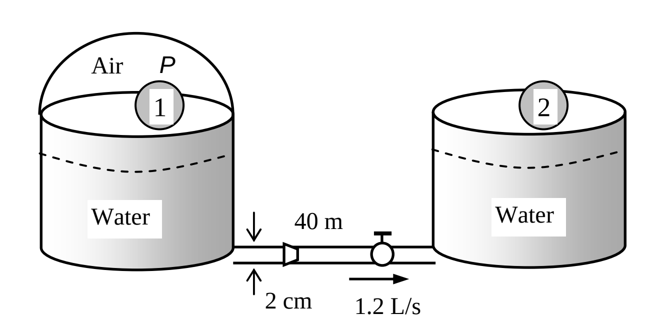
$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \;\rightarrow\; \frac{P_1}{\rho g}=\frac{P_{\text{atm}}}{\rho g}+h_L \;\rightarrow\; P_1=P_{\text{atm}}+\rho g h_L$$

where $h_L = h_{L,\text{total}} = h_{L,\text{major}} + h_{L,\text{minor}} = \left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}$

since the diameter of the piping system is constant. The average flow velocity and the Reynolds number are

$$V_2=\frac{\dot V}{A_c}=\frac{\dot V}{\pi D^2/4}=\frac{0.0012\ \text{m}^3/\text{s}}{\pi(0.02\ \text{m})^2/4}=3.82\ \text{m/s}$$

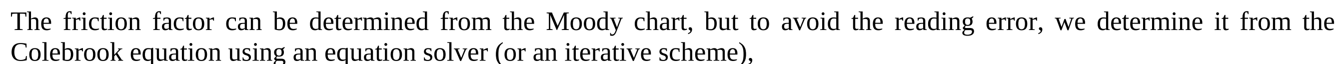
$$\text{Re}=\frac{\rho V_2 D}{\mu}=\frac{(999.7\ \text{kg/m}^3)(3.82\ \text{m/s})(0.02\ \text{m})}{1.307\times10^{-3}\ \text{kg/m}\cdot\text{s}}=58{,}400$$




which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D=\frac{0.00026\ \text{m}}{0.02\ \text{m}}=0.013$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),



$$\frac{1}{\sqrt f}=-2.0\log\left(\frac{\varepsilon/D}{3.7}+\frac{2.51}{\text{Re}\sqrt f}\right) \;\rightarrow\; \frac{1}{\sqrt f}=-2.0\log\left(\frac{0.013}{3.7}+\frac{2.51}{58{,}400\sqrt f}\right)$$

It gives $f$ = 0.0424. The sum of the loss coefficients is

$$\sum K_L = K_{L,\text{entrance}}+K_{L,\text{check valve}}+K_{L,\text{gate valve}}+K_{L,\text{exit}}=0.5+2+0.2+1=3.7$$

Then the total head loss becomes

$$h_L=\left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}=\left((0.0424)\frac{40\ \text{m}}{0.02\ \text{m}}+3.7\right)\frac{(3.82\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)}=65.8\ \text{m}$$

Substituting,

$$P_1=P_{\text{atm}}+\rho g h_L=(88\ \text{kPa})+(999.7\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(65.8\ \text{m})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{kPa}}{1\ \text{kN/m}^2}\right)=\mathbf{734\ kPa}$$

***Discussion*** The absolute pressure above the first reservoir must be 734 kPa, which is quite high. Note that the minor losses in this case are negligible (about 4% of total losses). Also, the friction factor could be determined easily from the explicit Haaland relation (it gives the same result, 0.0424). The friction coefficient would drop to 0.0202 if smooth pipes were used.

**8-80**

**Solution** A tanker is to be filled with fuel oil from an underground reservoir using a plastic hose. The required power input to the pump is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The flow is turbulent so that the tabulated value of the loss coefficients can be used (to be verified). **4** Fuel oil level remains constant. **5** Reservoir is open to the atmosphere.

***Properties*** The density and dynamic viscosity of fuel oil are given to be $\rho$ = 920 kg/m$^3$ and $\mu$ = 0.045 kg/m·s. The loss coefficient is $K_L$ = 0.12 for a slightly-rounded entrance and $K_L$ = 0.3 for a 90° smooth bend (flanged). The plastic pipe is smooth and thus $\varepsilon$ = 0. The kinetic energy correction factor at hose discharge is given to be $\alpha$ = 1.05.

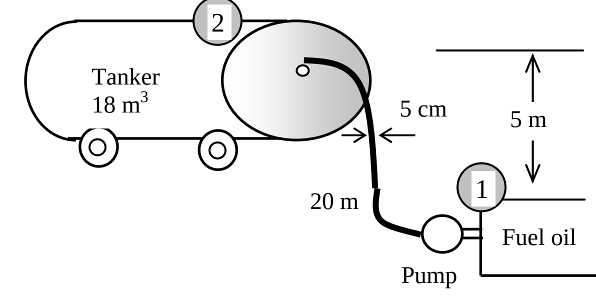
***Analysis*** We choose point 1 at the free surface of oil in the reservoir and point 2 at the exit of the hose in the tanker. We note the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and the fluid velocity at point 1 is zero ($V_1$ = 0). We take the free surface of the reservoir as the reference level ($z_1$ = 0). Then the energy equation for a control volume between these two points simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad h_{\text{pump, u}}=\alpha_2\frac{V_2^2}{2g}+z_2+h_L$$

where

$$h_L=h_{L,\text{total}}=h_{L,\text{major}}+h_{L,\text{minor}}=\left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}$$

since the diameter of the piping system is constant. The flow rate is determined from the requirement that the tanker must be filled in 30 min,




$$\dot{V}=\frac{V_{\text{tanker}}}{\Delta t}=\frac{18\text{ m}^3}{(30\times 60\text{ s})}=0.01\text{ m}^3/\text{s}$$

Then the average velocity in the pipe and the Reynolds number become

$$V_2=\frac{\dot{V}}{A_c}=\frac{\dot{V}}{\pi D^2/4}=\frac{0.01\text{ m}^3/\text{s}}{\pi(0.05\text{ m})^2/4}=5.093\text{ m/s}$$

$$\text{Re}=\frac{\rho V_2 D}{\mu}=\frac{(920\text{ kg/m}^3)(5.093\text{ m/s})(0.05\text{ m})}{0.045\text{ kg/m}\cdot\text{s}}=5206$$

which is greater than 4000. Therefore, the flow is turbulent. The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation,

$$\frac{1}{\sqrt{f}}=-2.0\log\left(\frac{\varepsilon/D}{3.7}+\frac{2.51}{\text{Re}\sqrt{f}}\right)\quad\rightarrow\quad\frac{1}{\sqrt{f}}=-2.0\log\left(0+\frac{2.51}{5206\sqrt{f}}\right)$$

It gives $f$ = 0.0370. The sum of the loss coefficients is

$$\sum K_L=K_{L,\text{entrance}}+2K_{L,\text{bend}}=0.12+2\times 0.3=0.72$$

Note that we do not consider the exit loss unless the exit velocity is dissipated within the system (in this case it is not). Then the total head loss, the useful pump head, and the required pumping power become

$$h_L=\left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}=\left((0.0370)\frac{20\text{ m}}{0.05\text{ m}}+0.72\right)\frac{(5.093\text{ m/s})^2}{2(9.81\text{ m/s}^2)}=20.5\text{ m}$$

$$h_{\text{pump, u}}=\frac{V_2^2}{2g}+z_2+h_L=1.05\frac{(5.093\text{ m/s})^2}{2(9.81\text{ m/s}^2)}+5\text{ m}+20.5\text{ m}=26.9\text{ m}$$

$$\dot{W}_{\text{pump}}=\frac{\dot{V}\rho g h_{\text{pump, u}}}{\eta_{\text{pump}}}=\frac{(0.01\text{ m}^3/\text{s})(920\text{ kg/m}^3)(9.81\text{ m/s}^2)(26.9\text{ m})}{0.82}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}}\right)=\mathbf{2.96\text{ kW}}$$

***Discussion*** Note that the minor losses in this case are negligible (0.72/15.52 = 0.046 or about 5% of total losses). Also, the friction factor could be determined easily from the Haaland relation (it gives 0.0372).

**8-81**

**Solution** Two pipes of identical length and material are connected in parallel. The diameter of one of the pipes is twice the diameter of the other. The ratio of the flow rates in the two pipes is to be determined

***Assumptions*** **1** The flow is steady and incompressible. **2** The friction factor is given to be the same for both pipes. **3** The minor losses are negligible.

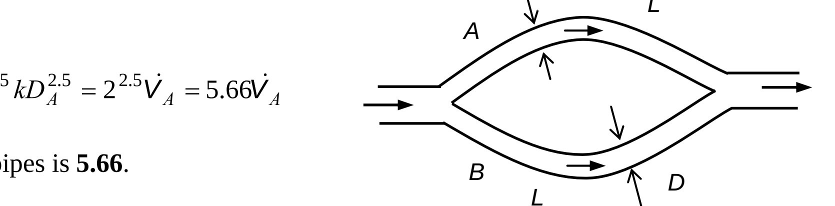
***Analysis*** When two pipes are parallel in a piping system, the head loss for each pipe must be same. When the minor losses are disregarded, the head loss for fully developed flow in a pipe of length $L$ and diameter $D$ can be expressed as

$$h_L = f\frac{L}{D}\frac{V^2}{2g} = f\frac{L}{D}\frac{1}{2g}\left(\frac{\dot{V}}{A_c}\right)^2 = f\frac{L}{D}\frac{1}{2g}\left(\frac{\dot{V}}{\pi D^2/4}\right)^2 = 8f\frac{L}{D}\frac{1}{g}\frac{\dot{V}^2}{\pi^2 D^4} = 8f\frac{L}{g\pi^2}\frac{\dot{V}^2}{D^5}$$

Solving for the flow rate gives

$$\dot{V} = \sqrt{\frac{\pi^2 h_L g}{8fL}}D^{2.5} = kD^{2.5} \qquad (k = \text{constant of proportionality})$$

When the pipe length, friction factor, and the head loss is constant, which is the case here for parallel connection, the flow rate becomes proportional to the $2.5^{th}$ power of diameter. Therefore, when the diameter is doubled, the flow rate will increase by a factor of $2^{2.5} = 5.66$ since




If $\qquad \dot{V}_A = kD_A^{2.5}$

Then $\qquad \dot{V}_B = kD_B^{2.5} = k(2D_A)^{2.5} = 2^{2.5}kD_A^{2.5} = 2^{2.5}\dot{V}_A = 5.66\dot{V}_A$

Therefore, the ratio of the flow rates in the two pipes is **5.66**.

***Discussion*** The relationship of flow rate to pipe diameter is not linear or even quadratic.

**8-82**

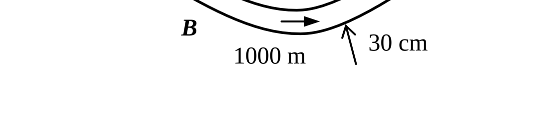
**Solution** Cast iron piping of a water distribution system involves a parallel section with identical diameters but different lengths. The flow rate through one of the pipes is given, and the flow rate through the other pipe is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The minor losses are negligible. **4** The flow is fully turbulent and thus the friction factor is independent of the Reynolds number (to be verified).

***Properties*** The density and dynamic viscosity of water at 15°C are $\rho$ = 999.1 kg/m$^3$ and $\mu$ = 1.138×10$^{-3}$ kg/m·s. The roughness of cast iron pipe is $\varepsilon$ = 0.00026 m.

***Analysis*** The average velocity in pipe *A* is

$$V_A = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.4\ \text{m}^3/\text{s}}{\pi(0.30\ \text{m})^2/4} = 5.659\ \text{m/s}$$




When two pipes are parallel in a piping system, the head loss for each pipe must be same. When the minor losses are disregarded, the head loss for fully developed flow in a pipe of length *L* and diameter *D* is

$$h_L = f\frac{L}{D}\frac{V^2}{2g}$$

Writing this for both pipes and setting them equal to each other, and noting that $D_A = D_B$ (given) and $f_A = f_B$ (to be verified) gives

$$f_A\frac{L_A}{D_A}\frac{V_A^2}{2g} = f_B\frac{L_B}{D_B}\frac{V_B^2}{2g} \quad\rightarrow\quad V_B = V_A\sqrt{\frac{L_A}{L_B}} = (5.659\ \text{m/s})\sqrt{\frac{1000\ \text{m}}{3000\ \text{m}}} = 3.267\ \text{m/s}$$

Then the flow rate in pipe *B* becomes

$$\dot{V}_B = A_c V_B = [\pi D^2/4]V_B = [\pi(0.30\ \text{m})^2/4](3.267\ \text{m/s}) = \mathbf{0.231\ m^3/s}$$

***Proof** that flow is fully turbulent and thus friction factor is independent of Reynolds number:*

The velocity in pipe *B* is lower. Therefore, if the flow is fully turbulent in pipe *B*, then it is also fully turbulent in pipe *A*. The Reynolds number in pipe *B* is

$$\text{Re}_B = \frac{\rho V_B D}{\mu} = \frac{(999.1\ \text{kg/m}^3)(3.267\ \text{m/s})(0.30\ \text{m})}{1.138\times10^{-3}\ \text{kg/m}\cdot\text{s}} = 0.860\times10^6$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D = \frac{0.00026\ \text{m}}{0.30\ \text{m}} = 0.00087$$

From Moody's chart, we observe that for a relative roughness of 0.00087, the flow is fully turbulent for Reynolds number greater than about $10^6$. Therefore, the flow in both pipes is fully turbulent, and thus the assumption that the friction factor is the same for both pipes is valid.

***Discussion*** Note that the flow rate in pipe *B* is less than the flow rate in pipe *A* because of the larger losses due to the larger length.

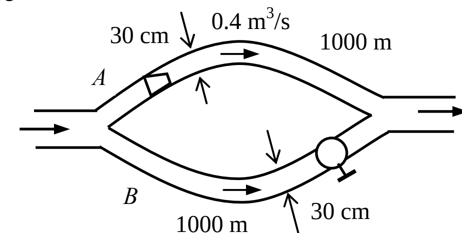
**8-83**

**Solution** Cast iron piping of a water distribution system involves a parallel section with identical diameters but different lengths and different valves. The flow rate through one of the pipes is given, and the flow rate through the other pipe is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The minor losses other than those for the valves are negligible. **4** The flow is fully turbulent and thus the friction factor is independent of the Reynolds number.

***Properties*** The density and dynamic viscosity of water at 15°C are $\rho$ = 999.1 kg/m$^3$ and $\mu$ = 1.138×10$^{-3}$ kg/m·s. The roughness of cast iron pipe is $\varepsilon$ = 0.00026 m.

***Analysis*** For pipe *A*, the average velocity and the Reynolds number are




$$V_A = \frac{\dot{V}_A}{A_c} = \frac{\dot{V}_A}{\pi D^2/4} = \frac{0.4\ \text{m}^3/\text{s}}{\pi(0.30\ \text{m})^2/4} = 5.659\ \text{m/s}$$

$$\text{Re}_A = \frac{\rho V_A D}{\mu} = \frac{(999.1\ \text{kg/m}^3)(5.659\ \text{m/s})(0.30\ \text{m})}{1.138\times10^{-3}\ \text{kg/m}\cdot\text{s}} = 1.49\times10^6$$

The relative roughness of the pipe is

$$\varepsilon/D = \frac{0.00026\ \text{m}}{0.30\ \text{m}} = 8.667\times10^{-4}$$

The friction factor corresponding to this relative roughness and the Reynolds number can simply be determined from the Moody chart. To avoid the reading error, we determine it from the Colebrook equation

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{8.667\times10^{-4}}{3.7} + \frac{2.51}{1.49\times10^6\sqrt{f}}\right)$$

It gives $f$ = 0.0192. Then the total head loss in pipe *A* becomes

$$h_{L,A} = \left(f\frac{L_A}{D} + K_L\right)\frac{V_A^2}{2g} = \left((0.0192)\frac{1000\ \text{m}}{0.30\ \text{m}} + 2.1\right)\frac{(5.659\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)} = 107.9\ \text{m}$$

When two pipes are parallel in a piping system, the head loss for each pipe must be same. Therefore, the head loss for pipe *B* must also be 107.9 m. Then the average velocity in pipe *B* and the flow rate become

$$h_{L,B} = \left(f\frac{L_B}{D} + K_L\right)\frac{V_B^2}{2g} \rightarrow 107.9\ \text{m} = \left((0.0192)\frac{3000\ \text{m}}{0.30\ \text{m}} + 10\right)\frac{V_B^2}{2(9.81\ \text{m/s}^2)} \rightarrow V_B = 3.24\ \text{m/s}$$

$$\dot{V}_B = A_c V_B = [\pi D^2/4]V_B = [\pi(0.30\ \text{m})^2/4](3.24\ \text{m/s}) = \mathbf{0.229\ m^3/s}$$

***Discussion*** Note that the flow rate in pipe *B* decreases slightly (from 0.231 to 0.229 m$^3$/s) due to the larger minor loss in that pipe. Also, minor losses constitute just a few percent of the total loss, and they can be neglected if great accuracy is not required.

**8-84**

**Solution** Geothermal water is supplied to a city through stainless steel pipes at a specified rate. The electric power consumption and its daily cost are to be determined, and it is to be assessed if the frictional heating during flow can make up for the temperature drop caused by heat loss.

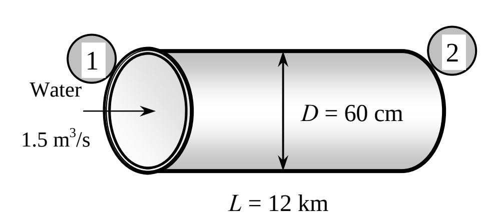
***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The minor losses are negligible because of the large length-to-diameter ratio and the relatively small number of components that cause minor losses. **4** The geothermal well and the city are at about the same elevation. **5** The properties of geothermal water are the same as fresh water. **6** The fluid pressures at the wellhead and the arrival point in the city are the same.

***Properties*** The properties of water at 110°C are $\rho$ = 950.6 kg/m$^3$, $\mu$ = 0.255×10$^{-3}$ kg/m·s, and $C_p$ = 4.229 kJ/kg·°C. The roughness of stainless steel pipes is 2×10$^{-6}$ m.

***Analysis*** (*a*) We take point 1 at the well-head of geothermal resource and point 2 at the final point of delivery at the city, and the entire piping system as the control volume. Both points are at the same elevation ($z_2 = z_2$) and the same velocity ($V_1 = V_2$) since the pipe diameter is constant, and the same pressure ($P_1 = P_2$). Then the energy equation for this control volume simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad h_{\text{pump, u}}=h_L$$

That is, the pumping power is to be used to overcome the head losses due to friction in flow. The average velocity and the Reynolds number are




$$V=\frac{\dot V}{A_c}=\frac{\dot V}{\pi D^2/4}=\frac{1.5\text{ m}^3/\text{s}}{\pi(0.60\text{ m})^2/4}=5.305\text{ m/s}$$

$$\text{Re}=\frac{\rho VD}{\mu}=\frac{(950.6\text{ kg/m}^3)(5.305\text{ m/s})(0.60\text{ m})}{0.255\times10^{-3}\text{ kg/m}\cdot\text{s}}=1.187\times10^7$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D=\frac{2\times10^{-6}\text{ m}}{0.60\text{ m}}=3.33\times10^{-6}$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt f}=-2.0\log\left(\frac{\varepsilon/D}{3.7}+\frac{2.51}{\text{Re}\sqrt f}\right)\rightarrow\frac{1}{\sqrt f}=-2.0\log\left(\frac{3.33\times10^{-6}}{3.7}+\frac{2.51}{1.187\times10^7\sqrt f}\right)$$

It gives $f$ = 0.00829. Then the pressure drop, the head loss, and the required power input become

$$\Delta P=\Delta P_L=f\frac{L}{D}\frac{\rho V^2}{2}=0.00829\frac{12{,}000\text{ m}}{0.60\text{ m}}\frac{(950.6\text{ kg/m}^3)(5.305\text{ m/s})^2}{2}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right)=2218\text{ kPa}$$

$$h_L=\frac{\Delta P_L}{\rho g}=f\frac{L}{D}\frac{V^2}{2g}=(0.00829)\frac{12{,}000\text{ m}}{0.60\text{ m}}\frac{(5.305\text{ m/s})^2}{2(9.81\text{ m/s}^2)}=238\text{ m}$$

$$\dot W_{\text{electric, in}}=\frac{\dot W_{\text{pump, u}}}{\eta_{\text{pump-motor}}}=\frac{\dot V\Delta P}{\eta_{\text{pump-motor}}}=\frac{(1.5\text{ m}^3/\text{s})(2218\text{ kPa})}{0.74}\left(\frac{1\text{ kW}}{1\text{ kPa}\cdot\text{m}^3/\text{s}}\right)=4496\text{ kW}\cong\mathbf{4500\text{ kW}}$$

Therefore, the pumps will consume 4496 kW of electric power to overcome friction and maintain flow. The pumps must raise the pressure of the geothermal water by 2218 kPa. Providing a pressure rise of this magnitude at one location may create excessive stress in piping at that location. Therefore, it is more desirable to raise the pressure by smaller amounts at a several locations along the flow. This will keep the maximum pressure in the system and the stress in piping at a safer level.

(*b*) The daily cost of electric power consumption is determined by multiplying the amount of power used per day by the unit cost of electricity,

$$\text{Amount}=\dot W_{\text{elect,in}}\Delta t=(4496\text{ kW})(24\text{ h/day})=107{,}900\text{ kWh/day}$$

$$\text{Cost}=\text{Amount}\times\text{Unit cost}=(107{,}900\text{ kWh/day})(\$0.06/\text{kWh})=\$6474/\text{day}\cong\mathbf{\$6470/day}$$

(*c*) The energy consumed by the pump (except the heat dissipated by the motor to the air) is eventually dissipated as heat due to the frictional effects. Therefore, this problem is equivalent to heating the water by a 4496 kW of resistance heater (again except the heat dissipated by the motor). To be conservative, we consider only the useful mechanical energy supplied to the water by the pump. The temperature rise of water due to this addition of energy is

$$\dot{W}_{\text{mech}} = \rho \dot{V} c_p \Delta T \;\rightarrow\; \Delta T = \frac{\eta_{\text{pump-motor}} \dot{W}_{\text{elect,in}}}{\rho \dot{V} c_p} = \frac{0.74 \times (4496 \text{ kJ/s})}{(950.6 \text{ kg/m}^3)(1.5 \text{ m}^3\text{/s})(4.229 \text{ kJ/kg}\cdot{}^\circ\text{C})} = \mathbf{0.55^\circ C}$$

Therefore, the temperature of water will rise at least 0.55°C, which is more than the 0.5°C drop in temperature (in reality, the temperature rise will be more since the energy dissipation due to pump inefficiency will also appear as temperature rise of water). Thus we conclude that the frictional heating during flow can more than make up for the temperature drop caused by heat loss.

***Discussion*** The pumping power requirement and the associated cost can be reduced by using a larger diameter pipe. But the cost savings should be compared to the increased cost of larger diameter pipe.

**8-85**

**Solution** Geothermal water is supplied to a city through cast iron pipes at a specified rate. The electric power consumption and its daily cost are to be determined, and it is to be assessed if the frictional heating during flow can make up for the temperature drop caused by heat loss.

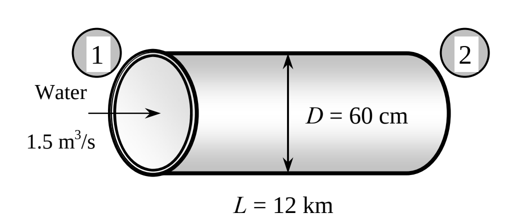
***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The minor losses are negligible because of the large length-to-diameter ratio and the relatively small number of components that cause minor losses. **4** The geothermal well and the city are at about the same elevation. **5** The properties of geothermal water are the same as fresh water. **6** The fluid pressures at the wellhead and the arrival point in the city are the same.

***Properties*** The properties of water at 110°C are $\rho$ = 950.6 kg/m$^3$, $\mu$ = 0.255×10$^{-3}$ kg/m·s, and $C_p$ = 4.229 kJ/kg·°C. The roughness of cast iron pipes is 0.00026 m.

***Analysis*** (*a*) We take point 1 at the well-head of geothermal resource and point 2 at the final point of delivery at the city, and the entire piping system as the control volume. Both points are at the same elevation ($z_2 = z_2$) and the same velocity ($V_1 = V_2$) since the pipe diameter is constant, and the same pressure ($P_1 = P_2$). Then the energy equation for this control volume simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad h_{\text{pump, u}}=h_L$$

That is, the pumping power is to be used to overcome the head losses due to friction in flow. The average velocity and the Reynolds number are




$$V=\frac{\dot V}{A_c}=\frac{\dot V}{\pi D^2/4}=\frac{1.5\text{ m}^3/\text{s}}{\pi(0.60\text{ m})^2/4}=5.305\text{ m/s}$$

$$\text{Re}=\frac{\rho VD}{\mu}=\frac{(950.6\text{ kg/m}^3)(5.305\text{ m/s})(0.60\text{ m})}{0.255\times10^{-3}\text{ kg/m}\cdot\text{s}}=1.187\times10^7$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D=\frac{0.00026\text{ m}}{0.60\text{ m}}=4.33\times10^{-4}$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt f}=-2.0\log\left(\frac{\varepsilon/D}{3.7}+\frac{2.51}{\text{Re}\sqrt f}\right)\rightarrow\frac{1}{\sqrt f}=-2.0\log\left(\frac{4.33\times10^{-4}}{3.7}+\frac{2.51}{1.187\times10^7\sqrt f}\right)$$

It gives $f$ = 0.0162. Then the pressure drop, the head loss, and the required power input become

$$\Delta P=\Delta P_L=f\frac{L}{D}\frac{\rho V^2}{2}=0.0162\frac{12{,}000\text{ m}}{0.60\text{ m}}\frac{(950.6\text{ kg/m}^3)(5.305\text{ m/s})^2}{2}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right)=4334\text{ kPa}$$

$$h_L=\frac{\Delta P_L}{\rho g}=f\frac{L}{D}\frac{V^2}{2g}=(0.0162)\frac{12{,}000\text{ m}}{0.60\text{ m}}\frac{(5.305\text{ m/s})^2}{2(9.81\text{ m/s}^2)}=465\text{ m}$$

$$\dot W_{\text{elect, in}}=\frac{\dot W_{\text{pump, u}}}{\eta_{\text{pump-motor}}}=\frac{\dot V\Delta P}{\eta_{\text{pump-motor}}}=\frac{(1.5\text{ m}^3/\text{s})(4334\text{ kPa})}{0.74}\left(\frac{1\text{ kW}}{1\text{ kPa}\cdot\text{m}^3/\text{s}}\right)=8785\text{ kW}\cong\mathbf{8790\text{ kW}}$$

Therefore, the pumps will consume 8785 kW of electric power to overcome friction and maintain flow. The pumps must raise the pressure of the geothermal water by 4334 kPa. Providing a pressure rise of this magnitude at one location may create excessive stress in piping at that location. Therefore, it is more desirable to raise the pressure by smaller amounts at a several locations along the flow. This will keep the maximum pressure in the system and the stress in piping at a safer level.

(*b*) The daily cost of electric power consumption is determined by multiplying the amount of power used per day by the unit cost of electricity,

$$\text{Amount}=\dot W_{\text{elect,in}}\Delta t=(8785\text{ kW})(24\text{ h/day})=210{,}800\text{ kWh/day}$$

$$\text{Cost}=\text{Amount}\times\text{Unit cost}=(210{,}800\text{ kWh/day})(\$0.06/\text{kWh})=\$12{,}650/\text{day}\cong\mathbf{\$12{,}700/day}$$

(*c*) The energy consumed by the pump (except the heat dissipated by the motor to the air) is eventually dissipated as heat

due to the frictional effects. Therefore, this problem is equivalent to heating the water by a 8785 kW of resistance heater (again except the heat dissipated by the motor). To be conservative, we consider only the useful mechanical energy supplied to the water by the pump. The temperature rise of water due to this addition of energy is

$$\dot{W}_{\text{mech}} = \rho \dot{V} c_p \Delta T \;\rightarrow\; \Delta T = \frac{\eta_{\text{pump-motor}} \dot{W}_{\text{elect,in}}}{\rho \dot{V} c_p} = \frac{0.74 \times (8785 \text{ kJ/s})}{(950.6 \text{ kg/m}^3)(1.5 \text{ m}^3\text{/s})(4.229 \text{ kJ/kg} \cdot {}^\circ\text{C})} = \mathbf{1.1^\circ C}$$

Therefore, the temperature of water will rise at least 1.1°C, which is more than the 0.5°C drop in temperature (in reality, the temperature rise will be more since the energy dissipation due to pump inefficiency will also appear as temperature rise of water). Thus we conclude that the frictional heating during flow can more than make up for the temperature drop caused by heat loss.

***Discussion*** The pumping power requirement and the associated cost can be reduced by using a larger diameter pipe. But the cost savings should be compared to the increased cost of larger diameter pipe.

**8-86E**

**Solution** The air discharge rate of a clothes drier with no ducts is given. The flow rate when duct work is attached is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects in the duct are negligible, and thus the flow is fully developed. **3** The flow is turbulent so that the tabulated value of the loss coefficients can be used. **4** The losses at the vent and its proximity are negligible. **5** The effect of the kinetic energy correction factor on discharge stream is negligible, $\alpha = 1$.

***Properties*** The density of air at 1 atm and 120°F is $\rho = 0.06843$ lbm/ft$^3$. The roughness of galvanized iron pipe is $\varepsilon = 0.0005$ ft. The loss coefficient is $K_L \approx 0$ for a well-rounded entrance with negligible loss, $K_L = 0.3$ for a flanged 90° smooth bend, and $K_L = 1.0$ for an exit. The friction factor of the duct is given to be 0.019.

***Analysis*** To determine the useful fan power input, we choose point 1 inside the drier sufficiently far from the vent, and point 2 at the exit on the same horizontal level so that $z_1 = z_2$ and $P_1 = P_2$, and the flow velocity at point 1 is negligible ($V_1 = 0$) since it is far from the inlet of the fan. Also, the frictional piping losses between 1 and 2 are negligible, and the only loss involved is due to fan inefficiency. Then the energy equation for a control volume between 1 and 2 reduces to

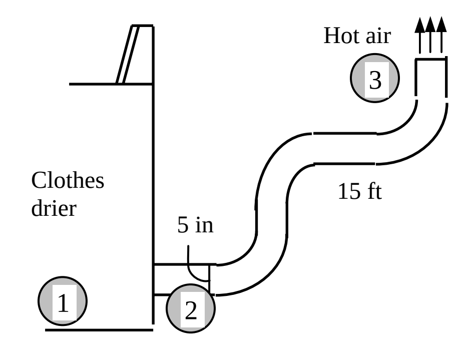
$$\dot{m}\left(\frac{P_1}{\rho}+\alpha_1\frac{V_1^2}{2}+gz_1\right)+\dot{W}_{\text{fan}}=\dot{m}\left(\frac{P_2}{\rho}+\alpha_2\frac{V_2^2}{2}+gz_2\right)+\dot{W}_{\text{turbine}}+\dot{E}_{\text{mech,loss}} \quad\rightarrow\quad \dot{W}_{\text{fan, u}}=\dot{m}\frac{V_2^2}{2} \qquad (1)$$

since $\alpha = 1$ and $\dot{E}_{\text{mech, loss}} = \dot{E}_{\text{mech loss, fan}} + \dot{E}_{\text{mech loss, piping}}$ and $\dot{W}_{\text{fan, u}} = \dot{W}_{\text{fan}} - \dot{E}_{\text{mech loss, fan}}$.

The average velocity is $V_2 = \dfrac{\dot{V}}{A_c} = \dfrac{\dot{V}}{\pi D^2/4} = \dfrac{1.2\ \text{ft}^3/\text{s}}{\pi(5/12\ \text{ft})^2/4} = 8.80$ ft/s

Now we attach the ductwork, and take point 3 to be at the duct exit so that the duct is included in the control volume. The energy equation for this control volume simplifies to

$$\dot{W}_{\text{fan, u}} = \dot{m}\frac{V_3^2}{2} + \dot{m}gh_L \qquad (2)$$




Combining (1) and (2),

$$\rho\dot{V}_2\frac{V_2^2}{2} = \rho\dot{V}_3\frac{V_3^2}{2} + \rho\dot{V}_3 gh_L \quad\rightarrow\quad \dot{V}_2\frac{V_2^2}{2} = \dot{V}_3\frac{V_3^2}{2} + \dot{V}_3 gh_L \qquad (3)$$

where

$$V_3 = \frac{\dot{V}_3}{A_c} = \frac{\dot{V}_3}{\pi D^2/4} = \frac{\dot{V}_3\ \text{ft}^3/\text{s}}{\pi(5/12\ \text{ft})^2/4} = 7.33\dot{V}_3\ \text{ft/s}$$

$$h_L = \left(f\frac{L}{D} + \sum K_L\right)\frac{V_3^2}{2g} = \left(0.019\frac{15\ \text{ft}}{5/12\ \text{ft}} + 3\times0.3 + 1\right)\frac{V_3^2}{2g} = 2.58\frac{V_3^2}{2g}$$

Substituting into Eq. (3),

$$\dot{V}_2\frac{V_2^2}{2} = \dot{V}_3\frac{V_3^2}{2} + \dot{V}_3 g\times2.58\frac{V_3^2}{2g} = \dot{V}_3\frac{(7.33\dot{V}_3)^2}{2} + \dot{V}_3\times2.58\frac{(7.33\dot{V}_3)^2}{2} = 96.2\dot{V}_3^3$$

Solving for $\dot{V}_3$ and substituting the numerical values gives

$$\dot{V}_3 = \left(\dot{V}_2\frac{V_2^2}{2\times96.2}\right)^{1/3} = \left(1.2\frac{8.80^2}{2\times96.2}\right)^{1/3} = \mathbf{0.78\ ft^3/s}$$

***Discussion*** Note that the flow rate decreased considerably for the same fan power input, as expected. We could also solve this problem by solving for the useful fan power first,

$$\dot{W}_{\text{fan, u}} = \rho\dot{V}_2\frac{V_2^2}{2} = (0.06843\ \text{lbm/ft}^3)(1.2\ \text{ft}^3/\text{s})\frac{(8.80\ \text{ft/s})^2}{2}\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right)\left(\frac{1\ \text{W}}{0.737\ \text{lbf}\cdot\text{ft/s}}\right) = 0.13\ \text{W}$$

Therefore, the fan supplies 0.13 W of useful mechanical power when the drier is running.

**8-87**

**Solution** Hot water in a water tank is circulated through a loop made of cast iron pipes at a specified average velocity. The required power input for the recirculating pump is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is fully developed. **3** The flow is turbulent so that the tabulated value of the loss coefficients can be used (to be verified). **4** Minor losses other than those for elbows and valves are negligible.

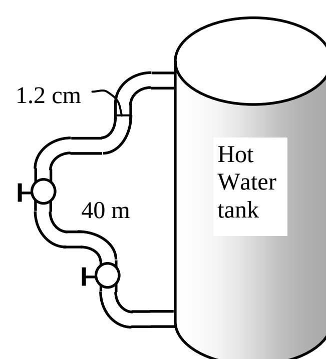
***Properties*** The density and dynamic viscosity of water at 60°C are $\rho$ = 983.3 kg/m$^3$, $\mu$ = 0.467×10$^{-3}$ kg/m·s. The roughness of cast iron pipes is 0.00026 m. The loss coefficient is $K_L$ = 0.9 for a threaded 90° smooth bend and $K_L$ = 0.2 for a fully open gate valve.

***Analysis*** Since the water circulates continually and undergoes a cycle, we can take the entire recirculating system as the control volume, and choose points 1 and 2 at any location at the same point. Then the properties (pressure, elevation, and velocity) at 1 and 2 will be identical, and the energy equation will simplify to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad h_{\text{pump, u}}=h_L$$

where

$$h_L=h_{L,\text{major}}+h_{L,\text{minor}}=\left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}$$




since the diameter of the piping system is constant. Therefore, the pumping power is to be used to overcome the head losses in the flow. The flow rate and the Reynolds number are

$$\dot V=VA_c=V(\pi D^2/4)=(2.5\text{ m/s})[\pi(0.012\text{ m})^2/4]=2.83\times10^{-4}\text{ m}^3\text{/s}$$

$$\text{Re}=\frac{\rho VD}{\mu}=\frac{(983.3\text{ kg/m}^3)(2.5\text{ m/s})(0.012\text{ m})}{0.467\times10^{-3}\text{ kg/m}\cdot\text{s}}=63{,}200$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D=\frac{0.00026\text{ m}}{0.012\text{ m}}=0.0217$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt f}=-2.0\log\left(\frac{\varepsilon/D}{3.7}+\frac{2.51}{\text{Re}\sqrt f}\right)\ \rightarrow\ \frac{1}{\sqrt f}=-2.0\log\left(\frac{0.0217}{3.7}+\frac{2.51}{63200\sqrt f}\right)$$

It gives $f$ = 0.05075. Then the total head loss, pressure drop, and the required pumping power input become

$$h_L=\left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}=\left((0.05075)\frac{40\text{ m}}{0.012\text{ m}}+6\times0.9+2\times0.2\right)\frac{(2.5\text{ m/s})^2}{2(9.81\text{ m/s}^2)}=55.8\text{ m}$$

$$\Delta P=\Delta P_L=\rho g h_L=(983.3\text{ kg/m}^3)(9.81\text{ m/s}^2)(55.8\text{ m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right)=538\text{ kPa}$$

$$\dot W_{\text{elect}}=\frac{\dot W_{\text{pump, u}}}{\eta_{\text{pump-motor}}}=\frac{\dot V\Delta P}{\eta_{\text{pump-motor}}}=\frac{(2.83\times10^{-4}\text{ m}^3\text{/s})(538\text{ kPa})}{0.70}\left(\frac{1\text{ kW}}{1\text{ kPa}\cdot\text{m}^3\text{/s}}\right)=\mathbf{0.217\ kW}$$

Therefore, **the required power input of the recirculating pump is 217 W**.

***Discussion*** It can be shown that the required pumping power input for the recirculating pump is 0.210 kW when the minor losses are not considered. Therefore, the minor losses can be neglected in this case without a major loss in accuracy.

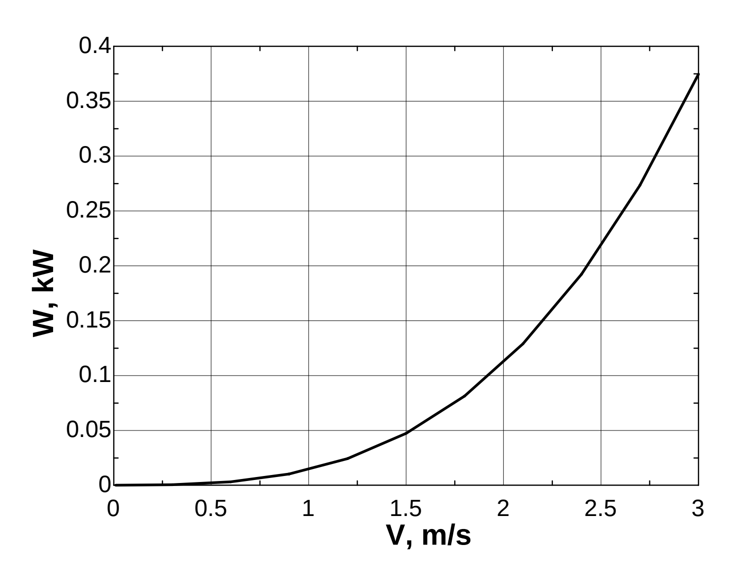

**8-88** 

**Solution** In the previous problem, the effect of average flow velocity on the power input to the recirculating pump for the same constant flow rate is to be investigated by varying the velocity from 0 to 3 m/s in increments of 0.3 m/s.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
g=9.81
rho=983.3
nu=mu/rho
mu=0.000467
D=0.012
L=40
KL=6*0.9+2*0.2
Eff=0.7
Ac=pi*D^2/4
Vdot=V*Ac
eps=0.00026
rf=eps/D
"Reynolds number"
Re=V*D/nu
1/sqrt(f)=-2*log10(rf/3.7+2.51/(Re*sqrt(f)))
DP=(f*L/D+KL)*rho*V^2/2000 "kPa"
W=Vdot*DP/Eff "kW"
HL=(f*L/D+KL)*(V^2/(2*g))
```

| $V$, m/s | $W_{pump}$, kW | $\Delta P_L$, kPa | Re |
|---|---|---|---|
| 0.0 | 0.0000 | 0.0 | 0 |
| 0.3 | 0.0004 | 8.3 | 7580 |
| 0.6 | 0.0031 | 32.0 | 15160 |
| 0.9 | 0.0103 | 71.0 | 22740 |
| 1.2 | 0.0243 | 125.3 | 30320 |
| 1.5 | 0.0472 | 195.0 | 37900 |
| 1.8 | 0.0814 | 279.9 | 45480 |
| 2.1 | 0.1290 | 380.1 | 53060 |
| 2.4 | 0.1922 | 495.7 | 60640 |
| 2.7 | 0.2733 | 626.6 | 68220 |
| 3.0 | 0.3746 | 772.8 | 75800 |




***Discussion*** As you might have suspected, the required power does not increase linearly with average velocity. Rather, the relationship is nearly quadratic. A larger diameter pipe would cut reduce the required pumping power considerably.

**8-89**

**Solution** Hot water in a water tank is circulated through a loop made of plastic pipes at a specified average velocity. The required power input for the recirculating pump is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is fully developed. **3** The flow is turbulent so that the tabulated value of the loss coefficients can be used (to be verified). **4** Minor losses other than those for elbows and valves are negligible.

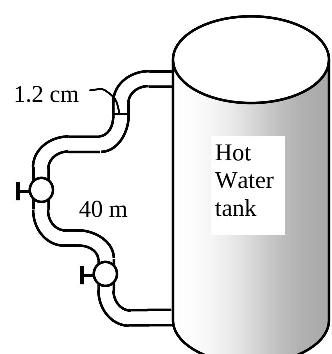
***Properties*** The density and viscosity of water at 60°C are $\rho$ = 983.3 kg/m$^3$, $\mu$ = 0.467×10$^{-3}$ kg/m·s. Plastic pipes are smooth, and thus their roughness is very close to zero, $\varepsilon$ = 0. The loss coefficient is $K_L$ = 0.9 for a threaded 90° smooth bend and $K_L$ = 0.2 for a fully open gate valve.

***Analysis*** Since the water circulates continually and undergoes a cycle, we can take the entire recirculating system as the control volume, and choose points 1 and 2 at any location at the same point. Then the properties (pressure, elevation, and velocity) at 1 and 2 will be identical, and the energy equation will simplify to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad h_{\text{pump, u}}=h_L$$

where

$$h_L=h_{L,\text{major}}+h_{L,\text{minor}}=\left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}$$




since the diameter of the piping system is constant. Therefore, the pumping power is to be used to overcome the head losses in the flow. The flow rate and the Reynolds number are

$$\dot V=VA_c=V(\pi D^2/4)=(2.5\text{ m/s})[\pi(0.012\text{ m})^2/4]=2.827\times10^{-4}\text{ m}^3\text{/s}$$

$$\text{Re}=\frac{\rho VD}{\mu}=\frac{(983.3\text{ kg/m}^3)(2.5\text{ m/s})(0.012\text{ m})}{0.467\times10^{-3}\text{ kg/m}\cdot\text{s}}=63{,}200$$

which is greater than 4000. Therefore, the flow is turbulent. The friction factor corresponding to the relative roughness of zero and this Reynolds number can simply be determined from the Moody chart. To avoid the reading error, we determine it from the Colebrook equation,

$$\frac{1}{\sqrt f}=-2.0\log\left(\frac{\varepsilon/D}{3.7}+\frac{2.51}{\text{Re}\sqrt f}\right)\quad\rightarrow\quad\frac{1}{\sqrt f}=-2.0\log\left(0+\frac{2.51}{63200\sqrt f}\right)$$

It gives $f$ = 0.0198. Then the total head loss, pressure drop, and the required pumping power input become

$$h_L=\left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}=\left((0.0198)\frac{40\text{ m}}{0.012\text{ m}}+6\times0.9+2\times0.2\right)\frac{(2.5\text{ m/s})^2}{2(9.81\text{ m/s}^2)}=22.9\text{ m}$$

$$\Delta P=\rho g h_L=(983.3\text{ kg/m}^3)(9.81\text{ m/s}^2)(22.9\text{ m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}}\right)\left(\frac{1\text{ kPa}}{1\text{ kN/m}^2}\right)=221\text{ kPa}$$

$$\dot W_{\text{elect}}=\frac{\dot W_{\text{pump, u}}}{\eta_{\text{pump-motor}}}=\frac{\dot V\Delta P}{\eta_{\text{pump-motor}}}=\frac{(2.827\times10^{-4}\text{ m}^3\text{/s})(221\text{ kPa})}{0.70}\left(\frac{1\text{ kW}}{1\text{ kPa}\cdot\text{m}^3\text{/s}}\right)=\mathbf{0.0893\ kW}$$

Therefore, **the required power input of the recirculating pump is 89.3 W**.

***Discussion*** It can be shown that the required pumping power input for the recirculating pump is 82.1 W when the minor losses are not considered. Therefore, the minor losses can be neglected in this case without a major loss in accuracy. Compared to the cast iron pipes of the previous problem, the plastic pipes reduced the required power by more than 50%, from 217 to 89.3 W. Furthermore, plastic pipes are lighter and easier to install, and they don't rust.

## Flow Rate and Velocity Measurements

### 8-90C

**Solution** We are to discus the primary considerations when choosing a flowmeter.

***Analysis*** The primary considerations when selecting a *flowmeter* are **cost, size, pressure drop, capacity, accuracy, and reliability**.

***Discussion*** As with just about everything you purchase, you usually get what you pay for.

### 8-91C

**Solution** We are to explain how a Pitot-static tube works and discuss its application.

***Analysis*** A *Pitot-static tube* **measures the difference between the stagnation and static pressure**, which is the *dynamic pressure*, which is related to flow velocity by $\boxed{V = \sqrt{2(P_1 - P_2)/\rho}}$. Once the average flow velocity is determined, the flow rate is calculated from $\dot{V} = VA_c$. The Pitot tube is inexpensive, highly reliable since it has no moving parts, it has very small pressure drop, and its accuracy (which is about 3%) is acceptable for most engineering applications.

***Discussion*** The term "Pitot tube" or "Pitot probe" is often used in place of "Pitot-static probe". Technically, however, a Pitot probe measures only stagnation pressure, while a Pitot-static probe measures both stagnation *and* static pressures.

### 8-92C

**Solution** We are to discuss the operation of obstruction flowmeters.

***Analysis*** An *obstruction flowmeter* measures the flow rate through a pipe by **constricting the flow, and measuring the decrease in pressure due to the increase in velocity at (or downstream of) the constriction site**. The flow rate for obstruction flowmeters is expressed as $\boxed{\dot{V} = A_o C_0 \sqrt{2(P_1 - P_2)/[\rho(1-\beta^4)]}}$ where $A_0 = \pi d^2/4$ is the cross-sectional area of the obstruction and $\beta = d/D$ is the ratio of obstruction diameter to the pipe diameter. Of the three types of obstruction flow meters, the orifice meter is the cheapest, smallest, and least accurate, and it causes the greatest head loss. The Venturi meter is the most expensive, the largest, the most accurate, and it causes the smallest head loss. The nozzle meter is between the orifice and Venturi meters in all aspects.

***Discussion*** As diameter ratio $\beta$ decreases, the pressure drop across the flowmeter increases, leading to a larger minor head loss associated with the flowmeter, but increasing the sensitivity of the measurement.

### 8-93C

**Solution** We are to discuss the operation of positive displacement flowmeters.

***Analysis*** A **positive displacement flowmeter** operates by **trapping a certain amount of incoming fluid, displacing it to the discharge side of the meter, and counting the number of such discharge-recharge cycles to determine the total amount of fluid displaced**. Positive displacement flowmeters are commonly used to meter gasoline, water, and natural gas because they are simple, reliable, inexpensive, and highly accurate even when the flow is unsteady.

***Discussion*** In applications such as a gasoline meter, it is not the flow *rate* that is measured, but the flow *volume*.

**8-94C**

**Solution** We are to discuss the operation of a turbine flowmeter.

***Analysis*** A *turbine flowmeter* consists of a **cylindrical flow section that houses a turbine that is free to rotate, and a sensor that generates a pulse each time a marked point on the turbine passes by to determine the rate of rotation**. Turbine flowmeters are relatively inexpensive, give highly accurate results (as accurate as 0.25%) over a wide range of flow rates, and cause a very small head loss.

***Discussion*** Turbine flowmeters must be calibrated so that a reading of the rpm of the turbine is translated into average velocity in the pipe or volume flow rate through the pipe.

---

**8-95C**

**Solution** We are to discuss the operation of rotameters.

***Analysis*** A *variable-area flowmeter* (or *rotameter*) consists of **a tapered conical transparent tube made of glass or plastic with a float inside that is free to move**. As fluid flows through the tapered tube, the float rises within the tube to a location where the float weight, drag force, and buoyancy force balance each other. Variable-area flowmeters are very simple devices with no moving parts except for the float (but even the float remains stationary during steady operation), and thus they are very reliable. They are also very inexpensive, and they cause a relatively small head loss.

***Discussion*** There are also some disadvantages. For example, they must be mounted vertically, and most of them require a visual reading, and so cannot be automated or connected to a computer system.

---

**8-96C**

**Solution** We are to compare thermal and laser Doppler anemometers.

***Analysis*** A *thermal anemometer* involves a **very small electrically heated sensor (hot wire) which loses heat to the fluid, and the flow velocity is related to the electric current needed to maintain the sensor at a constant temperature**. The flow velocity is determined by measuring the voltage applied or the electric current passing through the sensor. A *laser Doppler anemometer (LDA)* does not have a sensor that intrudes into flow. Instead, it **uses two laser beams that intersect at the point where the flow velocity is to be measured, and it makes use of the frequency shift (the Doppler effect) due to fluid flow to measure velocity**.

***Discussion*** Both of these devices measure the flow velocity at a point in the flow. Of the two, the hot wire system is much less expensive and has higher frequency resolution, but may interfere with the flow being measured.

---

**8-97C**

**Solution** We are to compare LDV and PIV.

***Analysis*** *Laser Doppler velocimetry (LDV)* **measures velocity at a point**, but *particle image velocimetry (PIV)* **provides velocity values simultaneously throughout an entire cross-section and thus it is a whole-field technique**. PIV combines the accuracy of LDV with the capability of flow visualization, and provides instantaneous flow field mapping. Both methods are non-intrusive, and both utilize laser light beams.

***Discussion*** In both cases, optical access is required – a hot-wire system does not require optical access, but, like the LDV system, measures velocity only at a single point.

---

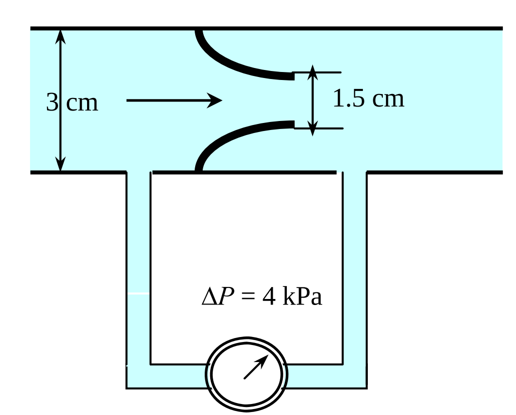
**8-98**

**Solution** The flow rate of ammonia is to be measured with flow nozzle equipped with a differential pressure gage. For a given pressure drop, the flow rate and the average flow velocity are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The discharge coefficient of the nozzle meter is $C_d = 0.96$.

***Properties*** The density and dynamic viscosity of ammonia are given to be $\rho = 624.6$ kg/m$^3$ and $\mu = 1.697\times10^{-4}$ kg/m·s, respectively.

***Analysis*** The diameter ratio and the throat area of the meter are




$$\beta = d/D = 1.5/3 = 0.50$$

$$A_0 = \pi d^2/4 = \pi(0.015\text{ m})^2/4 = 1.767\times10^{-4}\text{ m}^2$$

Noting that $\Delta P = 4$ kPa = 4000 N/m$^2$, the flow rate becomes

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}}$$

$$= (1.767\times10^{-4}\text{ m}^2)(0.96)\sqrt{\frac{2\times4000\text{ N/m}^2}{(624.6\text{ kg/m}^3)((1-0.50^4)}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)}$$

$$= \mathbf{0.627\times10^{-3}\ m^3/s}$$

which is equivalent to 0.627 L/s. The average flow velocity in the pipe is determined by dividing the flow rate by the cross-sectional area of the pipe,

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.627\times10^{-3}\text{ m}^3/\text{s}}{\pi(0.03\text{ m})^2/4} = \mathbf{0.887\ m/s}$$

***Discussion*** The Reynolds number of flow through the pipe is

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(624.6\text{ kg/m}^3)(0.887\text{ m/s})(0.03\text{ m})}{1.697\times10^{-4}\text{ kg/m}\cdot\text{s}} = 9.79\times10^4$$

Substituting the β and Re values into the orifice discharge coefficient relation gives

$$C_d = 0.9975 - \frac{6.53\beta^{0.5}}{\text{Re}^{0.5}} = 0.9975 - \frac{6.53(0.50)^{0.5}}{(9.79\times10^4)^{0.5}} = 0.983$$

which is about 2% different than the assumed value of 0.96. Using this refined value of $C_d$, the flow rate becomes 0.642 L/s, which differs from our original result by only 2.4%. If the problem is solved using an equation solver such as EES, then the problem can be formulated using the curve-fit formula for $C_d$ (which depends on Reynolds number), and all equations can be solved simultaneously by letting the equation solver perform the iterations as necessary.

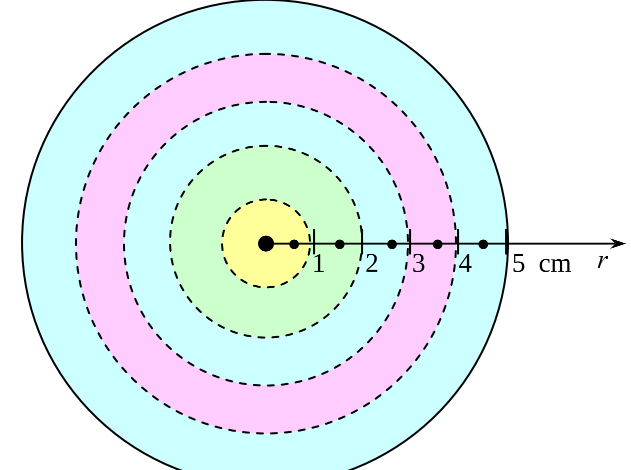
**8-99**

**Solution** The flow rate of water through a circular pipe is to be determined by measuring the water velocity at several locations along a cross-section. For a given set of measurements, the flow rate is to be determined.

***Assumptions*** The points of measurements are sufficiently close so that the variation of velocity between points can be assumed to be linear.

***Analysis*** The velocity measurements are given to be

| *R*, cm | *V*, m/s |
|---|---|
| 0 | 6.4 |
| 1 | 6.1 |
| 2 | 5.2 |
| 3 | 4.4 |
| 4 | 2.0 |
| 5 | 0.0 |




The divide the cross-section of the pipe into 1-cm thick annual regions, as shown in the figure. Using midpoint velocity values for each section, the flow rate is determined to be

$$\dot{V} = \int_{A_c} V dA_c \cong \sum V\pi(r_{out}^2 - r_{in}^2)$$

$$= \pi\left(\frac{6.4+6.1}{2}\right)(0.01^2 - 0) + \pi\left(\frac{6.1+5.2}{2}\right)(0.02^2 - 0.01^2) + \pi\left(\frac{5.2+4.4}{2}\right)(0.03^2 - 0.02^2)$$

$$+\pi\left(\frac{4.4+2.0}{2}\right)(0.04^2 - 0.02^2) + \pi\left(\frac{2.0+0}{2}\right)(0.05^2 - 0.04^2)$$

$$= \mathbf{0.0297\ m^3/s}$$

***Discussion*** We can also solve this problem by curve-fitting the given data using a second-degree polynomial, and then performing the integration.

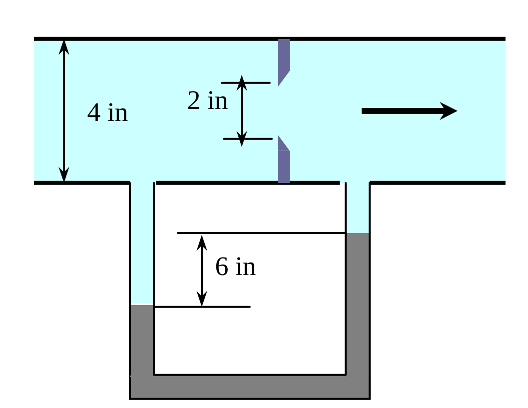
**8-100E**

**Solution** The flow rate of water is to be measured with an orifice meter. For a given pressure drop across the orifice plate, the flow rate, the average velocity, and the head loss are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The discharge coefficient of the orifice meter is $C_d = 0.61$.

***Properties*** The density and dynamic viscosity of water are given to be $\rho = 62.36$ kg/m$^3$ and $\mu = 7.536\times10^{-4}$ lbm/ft·s, respectively. We take the density of mercury to be 847 lbm/ft$^3$.

***Analysis*** The diameter ratio and the throat area of the orifice are




$$\beta = d/D = 2/4 = 0.50$$

$$A_0 = \pi d^2/4 = \pi(2/12\text{ ft})^2/4 = 0.02182\text{ ft}^2$$

The pressure drop across the orifice plate can be expressed as

$$\Delta P = P_1 - P_2 = (\rho_{\text{Hg}} - \rho_{\text{f}})gh$$

Then the flow rate relation for obstruction meters becomes

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1-P_2)}{\rho(1-\beta^4)}} = A_o C_d \sqrt{\frac{2(\rho_{\text{Hg}}-\rho_{\text{f}})gh}{\rho_{\text{f}}(1-\beta^4)}} = A_o C_d \sqrt{\frac{2(\rho_{\text{Hg}}/\rho_{\text{f}}-1)gh}{1-\beta^4}}$$

Substituting, the flow rate is determined to be

$$\dot{V} = (0.02182\text{ ft}^2)(0.61)\sqrt{\frac{2(847/62.36-1)(32.2\text{ ft/s}^2)(6/12\text{ ft})}{1-0.50^4}} = \mathbf{0.277\text{ ft}^3/s}$$

The average velocity in the pipe is determined by dividing the flow rate by the cross-sectional area of the pipe,

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.277\text{ ft}^3/\text{s}}{\pi(4/12\text{ ft})^2/4} = \mathbf{3.17\text{ ft/s}}$$

The percent pressure (or head) loss for orifice meters is given in Fig. 8-59 for $\beta = 0.5$ to be 74%. Therefore, noting that the density of mercury is 13.6 times that of water,

$$h_L = (\text{Permanent loss fraction})(\text{Total head loss}) = 0.74(0.50\text{ ft Hg}) = \mathbf{0.37\text{ ft Hg}} = \mathbf{5.03\text{ ft H}_2\text{O}}$$

The head loss between the two measurement sections can also be estimated from the energy equation. Since $z_1 = z_2$, the head form of the energy equation simplifies to

$$h_L \approx \frac{P_1-P_2}{\rho_f g} - \frac{V_2^2-V_1^2}{2g} = \frac{\rho_{Hg} g h_{Hg}}{\rho_f g} - \frac{\left[(D/d)^4-1\right]V_1^2}{2g}$$

$$= \text{SG}_{Hg} h_{Hg} - \frac{\left[(D/d)^4-1\right]V_1^2}{2g} = 13.6(0.50\text{ ft}) - \frac{\left[2^4-1\right](3.17\text{ ft/s})^2}{2(32.2\text{ ft/s}^2)} = \mathbf{4.46\text{ ft H}_2\text{O}}$$

This head loss, though a reasonable estimate, is lower than the exact one calculated above because it does not take into account irreversible losses downstream of the pressure taps, where the flow is still "recovering," and is not yet fully developed.

***Discussion*** The Reynolds number of flow through the pipe is

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(62.36\text{ kg/m}^3)(3.17\text{ ft/s})(4/12\text{ ft})}{7.536\times10^{-4}\text{ lbm/ft}\cdot\text{s}} = 8.744\times10^4$$

Substituting β and Re values into the orifice discharge coefficient relation

$$C_d = 0.5959 + 0.0312\beta^{2.1} - 0.184\beta^8 + \frac{91.71\beta^{2.5}}{\text{Re}^{0.75}}$$

gives $C_d = 0.606$, which is very close to the assumed value of 0.61. Using this refined value of $C_d$, the flow rate becomes 0.275 ft$^3$/s, which differs from our original result by less than 1%. Therefore, it is convenient to analyze orifice meters using the recommended value of $C_d = 0.61$ for the discharge coefficient, and then to verify the assumed value.

**8-101E**

**Solution** The flow rate of water is to be measured with an orifice meter. For a given pressure drop across the orifice plate, the flow rate, the average velocity, and the head loss are to be determined.

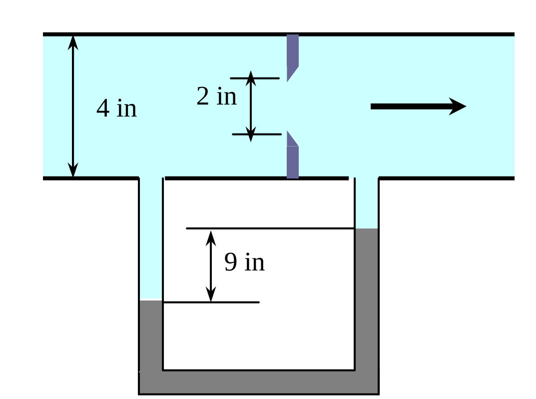
***Assumptions*** **1** The flow is steady and incompressible. **2** The discharge coefficient of the orifice meter is $C_d$ = 0.61.

***Properties*** The density and dynamic viscosity of water are given to be $\rho$ = 62.36 kg/m$^3$ and $\mu = 7.536\times10^{-4}$ lbm/ft·s, respectively. We take the density of mercury to be 847 lbm/ft$^3$.

***Analysis*** The diameter ratio and the throat area of the orifice are

$$\beta = d/D = 2/4 = 0.50$$

$$A_0 = \pi d^2/4 = \pi(2/12\text{ ft})^2/4 = 0.02182\text{ ft}^2$$




The pressure drop across the orifice plate can be expressed as

$$\Delta P = P_1 - P_2 = (\rho_{\text{Hg}} - \rho_{\text{f}})gh$$

Then the flow rate relation for obstruction meters becomes

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}} = A_o C_d \sqrt{\frac{2(\rho_{\text{Hg}} - \rho_{\text{f}})gh}{\rho_{\text{f}}(1-\beta^4)}} = A_o C_d \sqrt{\frac{2(\rho_{\text{Hg}}/\rho_{\text{f}} - 1)gh}{1-\beta^4}}$$

Substituting, the flow rate is determined to be

$$\dot{V} = (0.02182\text{ ft}^2)(0.61)\sqrt{\frac{2(847/62.36-1)(32.2\text{ ft/s}^2)(9/12\text{ ft})}{1-0.50^4}} = \mathbf{0.339\ ft^3/s}$$

The average velocity in the pipe is determined by dividing the flow rate by the cross-sectional area of the pipe,

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.339\text{ ft}^3/\text{s}}{\pi(4/12\text{ ft})^2/4} = \mathbf{3.88\ ft/s}$$

The percent pressure (or head) loss for orifice meters is given in Fig. 8-59 for $\beta$ = 0.5 to be 74%. Therefore, noting that the density of mercury is 13.6 times that of water,

$$h_L = \text{(Permanent loss fraction)(Total head loss)} = 0.74(0.75\text{ ft Hg}) = \mathbf{0.555\ ft\ Hg} = \mathbf{7.55\ ft\ H2O}$$

The head loss between the two measurement sections can also be estimated from the energy equation. Since $z_1 = z_2$, the head form of the energy equation simplifies to

$$h_L \approx \frac{P_1 - P_2}{\rho_f g} - \frac{V_2^2 - V_1^2}{2g} = \frac{\rho_{Hg} g h_{Hg}}{\rho_f g} - \frac{\left[(D/d)^4 - 1\right]V_1^2}{2g}$$

$$= \text{SG}_{Hg} h_{Hg} - \frac{\left[(D/d)^4 - 1\right]V_1^2}{2g} = 13.6(0.75\text{ ft}) - \frac{\left[2^4 - 1\right](3.88\text{ ft/s})^2}{2(32.2\text{ ft/s}^2)} = \mathbf{6.69\ ft\ H_2O}$$

This head loss, though a reasonable estimate, is lower than the exact one calculated above because it does not take into account irreversible losses downstream of the pressure taps, where the flow is still "recovering," and is not yet fully developed.

***Discussion*** The Reynolds number of flow through the pipe is

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(62.36\text{ kg/m}^3)(3.88\text{ ft/s})(4/12\text{ ft})}{7.536\times10^{-4}\text{ lbm/ft}\cdot\text{s}} = 1.070\times10^5$$

Substituting β and Re values into the orifice discharge coefficient relation

$$C_d = 0.5959 + 0.0312\beta^{2.1} - 0.184\beta^8 + \frac{91.71\beta^{2.5}}{\text{Re}^{0.75}}$$

gives $C_d$ = 0.605, which is very close to the assumed value of 0.61. Using this refined value of $C_d$, the flow rate becomes 0.336 ft$^3$/s, which differs from our original result by less than 1%. Therefore, it is convenient to analyze orifice meters using the recommended value of $C_d$ = 0.61 for the discharge coefficient, and then to verify the assumed value.

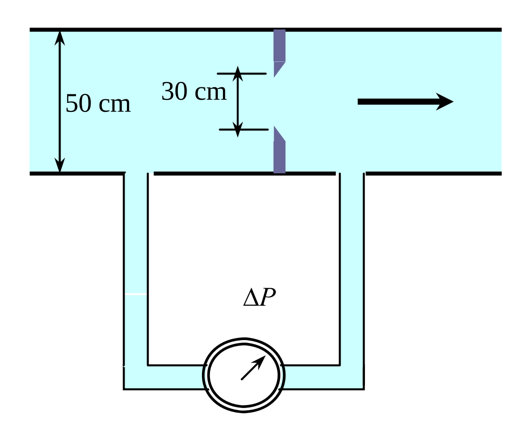
**8-102**

**Solution** The flow rate of water is measured with an orifice meter. The pressure difference indicated by the orifice meter and the head loss are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The discharge coefficient of the orifice meter is $C_d = 0.61$.

***Properties*** The density and dynamic viscosity of water are given to be $\rho$ = 998 kg/m$^3$ and $\mu$ = 1.002×10$^{-4}$ kg/m·s, respectively.

***Analysis*** The diameter ratio and the throat area of the orifice are




$$\beta = d / D = 30/50 = 0.60$$

$$A_0 = \pi d^2 / 4 = \pi(0.30 \text{ m})^2 / 4 = 0.07069 \text{ m}^2$$

For a pressure drop of $\Delta P = P_1 - P_2$ across the orifice plate, the flow rate is expressed as

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}}$$

Substituting,

$$0.25 \text{ m}^3/s = (0.07069 \text{ m}^2)(0.61)\sqrt{\frac{2\Delta P}{(998 \text{ kg/m}^3)((1-0.60^4)}}$$

which gives the pressure drop across the orifice plate to be

$$\Delta P = 14{,}600 \text{ kg}\cdot\text{m/s}^2 = \mathbf{14.6\ kPa}$$

It corresponds to a water column height of

$$h_w = \frac{\Delta P}{\rho_w g} = \frac{14{,}600 \text{ kg}\cdot\text{m/s}^2}{(998 \text{ kg/m}^3)(9.81 \text{ m/s}^2)} = 1.49 \text{ m}$$

The percent pressure (or head) loss for orifice meters is given in Fig. 8-59 for $\beta$ = 0.6 to be 64%. Therefore,



$$h_L = (\text{Permanent loss fraction})(\text{Total head loss}) = 0.64(1.49 \text{ m}) = \mathbf{0.95\ m\ H_2O}$$

The head loss between the two measurement sections can also be estimated from the energy equation. Since $z_1 = z_2$, the head form of the energy equation simplifies to

$$h_L \approx \frac{P_1 - P_2}{\rho_f g} - \frac{V_2^2 - V_1^2}{2g} = h_w - \frac{\left[(D/d)^4 - 1\right]V_1^2}{2g} = 1.49 \text{ m} - \frac{\left[(50/30)^4 - 1\right](1.27 \text{ m/s})^2}{2(9.81 \text{ m/s}^2)} = \mathbf{0.940\ m\ H_2O}$$

where $V_1 = \dfrac{\dot{V}}{A_c} = \dfrac{\dot{V}}{\pi D^2/4} = \dfrac{0.250 \text{ m}^3/\text{s}}{\pi(0.50 \text{ m})^2/4} = 1.27 \text{ m/s}$

This head loss, though a reasonable estimate, is lower than the exact one calculated above because it does not take into account irreversible losses downstream of the pressure taps, where the flow is still "recovering," and is not yet fully developed.

***Discussion*** The Reynolds number of flow through the pipe is

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(998 \text{ kg/m}^3)(1.27 \text{ m/s})(0.50 \text{ m})}{1.002\times10^{-3} \text{ kg/m}\cdot\text{s}} = 6.32\times10^5$$

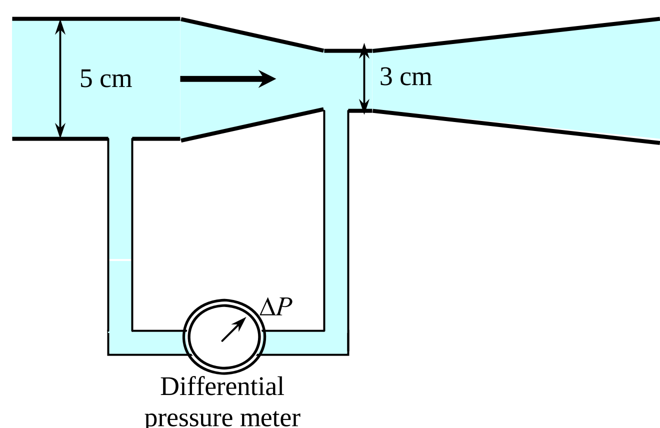
Substituting β and Re values into the orifice discharge coefficient relation

$$C_d = 0.5959 + 0.0312\beta^{2.1} - 0.184\beta^8 + \frac{91.71\beta^{2.5}}{\text{Re}^{0.75}}$$

gives $C_d$ = 0.605, which is very close to the assumed value of 0.61.

**8-103**

**Solution** A Venturi meter equipped with a differential pressure gage is used to measure to flow rate of water through a horizontal pipe. For a given pressure drop, the volume flow rate of water and the average velocity through the pipe are to be determined.




***Assumptions*** The flow is steady and incompressible.

***Properties*** The density of water is given to be $\rho$ = 999.1 kg/m$^3$. The discharge coefficient of Venturi meter is given to be $C_d$ = 0.98.

***Analysis*** The diameter ratio and the throat area of the meter are

$$\beta = d / D = 3/5 = 0.60$$

$$A_0 = \pi d^2 / 4 = \pi (0.03\,\text{m})^2 / 4 = 7.069\times10^{-4}\ \text{m}^2$$

Noting that $\Delta P$ = 5 kPa = 5000 N/m$^2$, the flow rate becomes

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}}$$

$$= (7.069\times10^{-4}\ \text{m}^2)(0.98)\sqrt{\frac{2\times5000\ \text{N/m}^2}{(999.1\,\text{kg/m}^3)((1-0.60^4)}\left(\frac{1\,\text{kg}\cdot\text{m/s}^2}{1\,\text{N}}\right)}$$

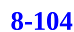
$$= \mathbf{0.00235\ m^3/s}$$

which is equivalent to 2.35 L/s. The average flow velocity in the pipe is determined by dividing the flow rate by the cross-sectional area of the pipe,

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.00235\,\text{m}^3/\text{s}}{\pi(0.05\,\text{m})^2/4} = \mathbf{1.20\ m/s}$$

***Discussion*** Note that the flow rate is proportional to the square root of pressure difference across the Venturi meter.

---



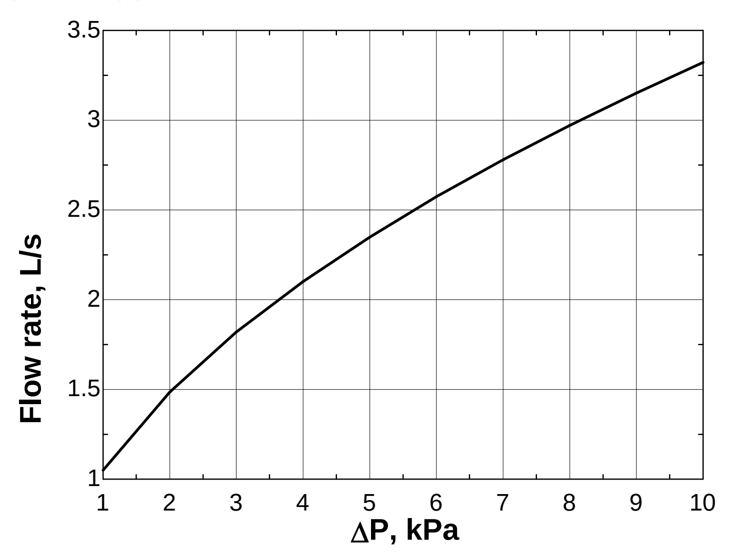
**8-104** 

**Solution** The previous problem is reconsidered. The variation of flow rate as the pressure drop varies from 1 kPa to 10 kPa at intervals of 1 kPa is to be investigated, and the results are to be plotted.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
rho=999.1 "kg/m3"
D=0.05 "m"
d0=0.03 "m"
beta=d0/D
A0=pi*d0^2/4
Cd=0.98
Vol=A0*Cd*SQRT(2*DeltaP*1000/(rho*(1-beta^4)))*1000 "L/s"
```

| Pressure Drop $\Delta P$, kPa | Flow rate L/s |
|---|---|
| 1 | 1.05 |
| 2 | 1.49 |
| 3 | 1.82 |
| 4 | 2.10 |
| 5 | 2.35 |
| 6 | 2.57 |
| 7 | 2.78 |
| 8 | 2.97 |
| 9 | 3.15 |
| 10 | 3.32 |

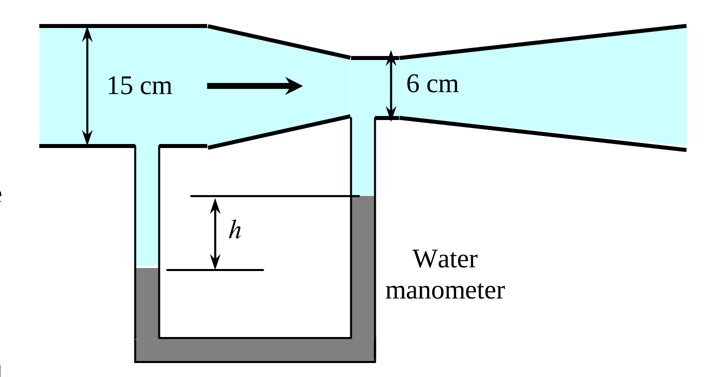



***Discussion*** This type of plot can be thought of as a calibration plot for the flowmeter, although a real calibration plot would use actual experimental data rather than data from equations. It would be interesting to compare the above plot to experimental data to see how close the predictions are.

**8-105**

**Solution** A Venturi meter equipped with a water manometer is used to measure to flow rate of air through a duct. For a specified maximum differential height for the manometer, the maximum mass flow rate of air that can be measured is to be determined.

***Assumptions*** The flow is steady and incompressible.




***Properties*** The density of air is given to be $\rho_{air}$ = 1.204 kg/m$^3$. We take the density of water to be $\rho_w$ = 1000 kg/m$^3$. The discharge coefficient of Venturi meter is given to be $C_d$ = 0.98.

***Analysis*** The diameter ratio and the throat area of the meter are

$$\beta = d / D = 6/15 = 0.40$$

$$A_0 = \pi d^2 / 4 = \pi (0.06\text{ m})^2 / 4 = 0.002827\text{ m}^2$$

The pressure drop across the Venturi meter can be expressed as

$$\Delta P = P_1 - P_2 = (\rho_w - \rho_f)gh$$

Then the flow rate relation for obstruction meters becomes

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}} = A_o C_d \sqrt{\frac{2(\rho_w - \rho_f)gh}{\rho_f(1-\beta^4)}} = A_o C_d \sqrt{\frac{2(\rho_w/\rho_{air} - 1)gh}{1-\beta^4}}$$

Substituting and using $h$ = 0.40 m, the maximum volume flow rate is determined to be

$$\dot{V} = (0.002827\text{ m}^2)(0.98)\sqrt{\frac{2(1000/1.204 - 1)(9.81\text{ m/s}^2)(0.40\text{ m})}{1-0.40^4}} = 0.2265\text{ m}^3/s$$

Then the maximum mass flow rate this Venturi meter can measure is

$$\dot{m} = \rho\dot{V} = (1.204\text{ kg/m}^3)(0.2265\text{ m}^3\text{/s}) = \mathbf{0.273\ kg/s}$$

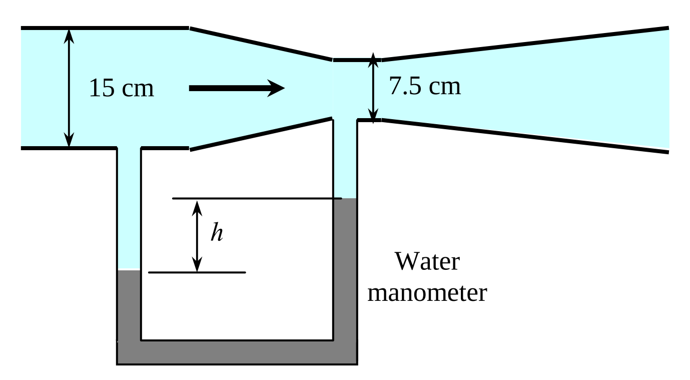
Also, the average flow velocity in the duct is

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.2265\text{ m}^3/\text{s}}{\pi(0.15\text{ m})^2/4} = 12.8\text{ m/s}$$

***Discussion*** Note that the maximum available differential height limits the flow rates that can be measured with a manometer.

**8-106**

**Solution** A Venturi meter equipped with a water manometer is used to measure to flow rate of air through a duct. For a specified maximum differential height for the manometer, the maximum mass flow rate of air that can be measured is to be determined.

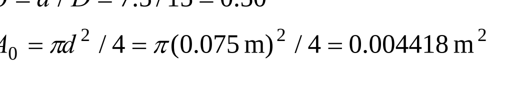



***Assumptions*** The flow is steady and incompressible.

***Properties*** The density of air is given to be $\rho_{air}$ = 1.204 kg/m$^3$. We take the density of water to be $\rho_w$ = 1000 kg/m$^3$. The discharge coefficient of Venturi meter is given to be $C_d$ = 0.98.

***Analysis*** The diameter ratio and the throat area of the meter are

$$\beta = d / D = 7.5/15 = 0.50$$



$$A_0 = \pi d^2/4 = \pi(0.075\,\text{m})^2/4 = 0.004418\,\text{m}^2$$

The pressure drop across the Venturi meter can be expressed as

$$\Delta P = P_1 - P_2 = (\rho_w - \rho_f)gh$$

Then the flow rate relation for obstruction meters becomes

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}} = A_o C_d \sqrt{\frac{2(\rho_w - \rho_f)gh}{\rho_f(1-\beta^4)}} = A_o C_d \sqrt{\frac{2(\rho_w/\rho_{air} - 1)gh}{1-\beta^4}}$$

Substituting and using $h$ = 0.40 m, the maximum volume flow rate is determined to be

$$\dot{V} = (0.004418\,\text{m}^2)(0.98)\sqrt{\frac{2(1000/1.204 - 1)(9.81\,\text{m/s}^2)(0.40\,\text{m})}{1 - 0.50^4}} = 0.3608\,\text{m}^3/s$$

Then the maximum mass flow rate this Venturi meter can measure is

$$\dot{m} = \rho\dot{V} = (1.204\,\text{kg/m}^3)(0.3608\,\text{m}^3\text{/s}) = \mathbf{0.434\ kg/s}$$

Also, the average flow velocity in the duct is

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.3608\,\text{m}^3/\text{s}}{\pi(0.15\,\text{m})^2/4} = 20.4\,\text{m/s}$$

***Discussion*** Note that the maximum available differential height limits the flow rates that can be measured with a manometer.

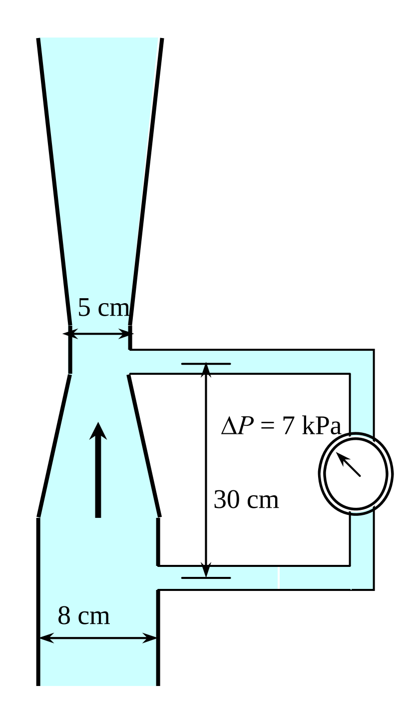
---

**8-107**

**Solution** A Venturi meter equipped with a differential pressure gage is used to measure the flow rate of liquid propane through a vertical pipe. For a given pressure drop, the volume flow rate is to be determined.

***Assumptions*** The flow is steady and incompressible.

***Properties*** The density of propane is given to be $\rho = 514.7$ kg/m$^3$. The discharge coefficient of Venturi meter is given to be $C_d = 0.98$.




***Analysis*** The diameter ratio and the throat area of the meter are

$$\beta = d/D = 5/8 = 0.625$$

$$A_0 = \pi d^2/4 = \pi(0.05\text{ m})^2/4 = 0.001963\text{ m}^2$$

Noting that $\Delta P = 7$ kPa = 7000 N/m$^2$, the flow rate becomes

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}}$$

$$= (0.001963\text{ m}^2)(0.98)\sqrt{\frac{2 \times 7000\text{ N/m}^2}{(514.7\text{ kg/m}^3)((1-0.625^4)}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)}$$

$$= \mathbf{0.0109\ m^3/s}$$

which is equivalent to 10.9 L/s. Also, the average flow velocity in the pipe is

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.0109\text{ m}^3/\text{s}}{\pi(0.08\text{ m})^2/4} = 2.17\text{ m/s}$$

***Discussion*** Note that the elevation difference between the locations of the two probes does not enter the analysis since the pressure gage measures the pressure differential at a specified location. When there is no flow through the Venturi meter, for example, the pressure gage would read zero.

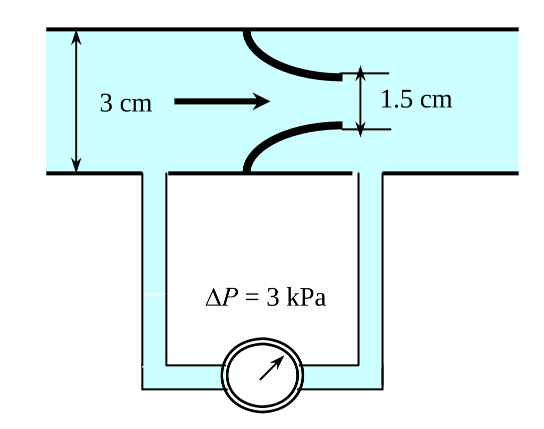
**8-108**

**Solution** The flow rate of water is to be measured with flow nozzle equipped with a differential pressure gage. For a given pressure drop, the flow rate, the average flow velocity, and head loss are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The discharge coefficient of the nozzle meter is $C_d = 0.96$.

***Properties*** The density and dynamic viscosity of water are given to be $\rho = 999.7$ kg/m$^3$ and $\mu = 1.307\times10^{-3}$ kg/m·s, respectively.

***Analysis*** The diameter ratio and the throat area of the meter are




$$\beta = d/D = 1.5/3 = 0.50$$

$$A_0 = \pi d^2/4 = \pi(0.015\text{ m})^2/4 = 1.767\times10^{-4}\text{ m}^2$$

Noting that $\Delta P$ = 4 kPa = 4000 N/m$^2$, the flow rate becomes

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}}$$

$$= (1.767\times10^{-4}\text{ m}^2)(0.96)\sqrt{\frac{2\times3000\text{ N/m}^2}{(999.7\text{ kg/m}^3)((1-0.50^4)}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)}$$

$$= \mathbf{0.429\times10^{-3}\ m^3/s}$$

which is equivalent to 0.429 L/s. The average flow velocity in the pipe is determined by dividing the flow rate by the cross-sectional area of the pipe,

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.429\times10^{-3}\text{ m}^3/\text{s}}{\pi(0.03\text{ m})^2/4} = \mathbf{0.607\ m/s}$$

The water column height corresponding to a pressure drop of 3 kPa is

$$h_w = \frac{\Delta P}{\rho_w g} = \frac{3000\text{ kg}\cdot\text{m/s}^2}{(999.7\text{ kg/m}^3)(9.81\text{ m/s}^2)} = 0.306\text{ m}$$

The percent pressure (or head) loss for nozzle meters is given in Fig. 8-59 for $\beta = 0.5$ to be 62%. Therefore,

$$h_L = (\text{Permanent loss fraction})(\text{Total head loss}) = 0.62(0.306\text{ m}) = \mathbf{0.19\ m\ H_2O}$$

The head loss between the two measurement sections can be determined from the energy equation, which simplifies to (for $z_1 = z_2$)

$$h_L = \frac{P_1 - P_2}{\rho_f g} - \frac{V_2^2 - V_1^2}{2g} = h_w - \frac{[(D/d)^4 - 1]V_1^2}{2g} = 0.306\text{ m} - \frac{[(3/1.5)^4 - 1](0.607\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 0.024\text{ m H2O}$$

***Discussion*** The Reynolds number of flow through the pipe is

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(999.7\text{ kg/m}^3)(0.607\text{ m/s})(0.03\text{ m})}{1.307\times10^{-3}\text{ kg/m}\cdot\text{s}} = 1.39\times10^4$$

Substituting the β and Re values into the orifice discharge coefficient relation gives

$$C_d = 0.9975 - \frac{6.53\beta^{0.5}}{\text{Re}^{0.5}} = 0.9975 - \frac{6.53(0.50)^{0.5}}{(1.39\times10^4)^{0.5}} = 0.958$$

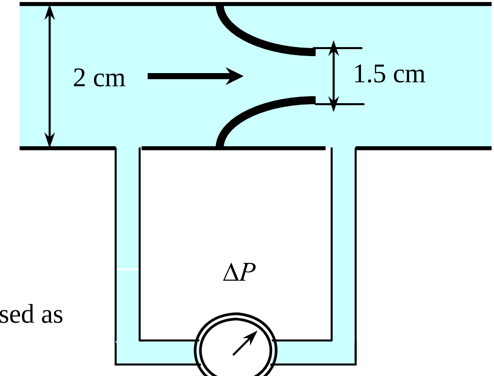
which is practically identical to the assumed value of 0.96.

**8-109**

**Solution** A kerosene tank is filled with a hose equipped with a nozzle meter. For a specified filling time, the pressure difference indicated by the nozzle meter is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The discharge coefficient of the nozzle meter is $C_d = 0.96$.

***Properties*** The density of kerosene is given to be $\rho = 820\ \text{kg/m}^3$.




***Analysis*** The diameter ratio and the throat area of the meter are

$$\beta = d / D = 1.5 / 2 = 0.75$$

$$A_0 = \pi d^2 / 4 = \pi (0.015\ \text{m})^2 / 4 = 1.767 \times 10^{-4}\ \text{m}^2$$

To fill a 16-L tank in 20 s, the flow rate must be

$$\dot{V} = \frac{V_{\text{tank}}}{\Delta t} = \frac{16\ \text{L}}{20\ \text{s}} = 0.8\ \text{L/s}$$

For a pressure drop of $\Delta P = P_1 - P_2$ across the meter, the flow rate is expressed as

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}}$$

Substituting,

$$0.0008\ \text{m}^3/s = (1.767 \times 10^{-4}\ \text{m}^2)(0.96)\sqrt{\frac{2\Delta P}{(820\ \text{kg/m}^3)((1-0.75^4)}}$$

which gives the pressure drop across the meter to be

$$\Delta P = 6230\ \text{kg}\cdot\text{m/s}^2 = \mathbf{6.23\ kPa}$$

***Discussion*** Note that the flow rate is proportional to the square root of pressure difference across the nozzle meter.

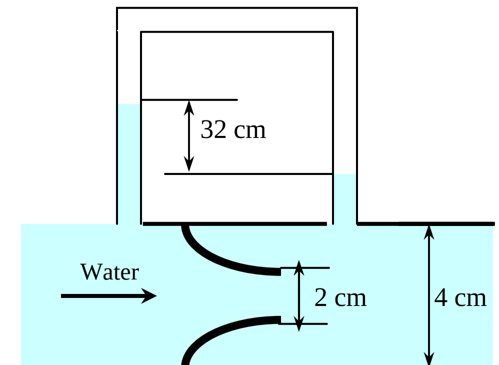
**8-110**

**Solution** The flow rate of water is to be measured with flow nozzle equipped with an inverted air-water manometer. For a given differential height, the flow rate and head loss caused by the nozzle meter are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The discharge coefficient of the nozzle meter is $C_d$ = 0.96.

***Properties*** The density and dynamic viscosity of water are given to be $\rho$ = 998 kg/m$^3$ and $\mu$ = 1.002×10$^{-3}$ kg/m·s, respectively.

***Analysis*** The diameter ratio and the throat area of the meter are

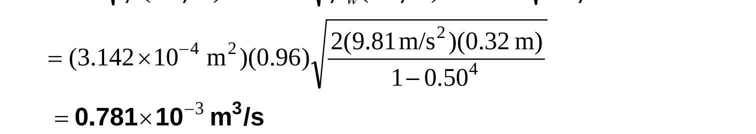



$$\beta = d/D = 2/4 = 0.50$$

$$A_0 = \pi d^2/4 = \pi(0.02\text{ m})^2/4 = 3.142\times10^{-4}\text{ m}^2$$

Noting that $\Delta P$ = 4 kPa = 4000 N/m$^2$, the flow rate becomes

$$\dot{V} = A_o C_d\sqrt{\frac{2(P_1-P_2)}{\rho(1-\beta^4)}} = A_o C_d\sqrt{\frac{2\rho_w gh}{\rho_w(1-\beta^4)}} = A_o C_d\sqrt{\frac{2gh}{1-\beta^4}}$$



$$= (3.142\times10^{-4}\text{ m}^2)(0.96)\sqrt{\frac{2(9.81\text{ m/s}^2)(0.32\text{ m})}{1-0.50^4}}$$

$$= \mathbf{0.781\times10^{-3}\ m^3/s}$$

which is equivalent to 0.781 L/s. The average flow velocity in the pipe is

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.781\times10^{-3}\text{ m}^3/\text{s}}{\pi(0.04\text{ m})^2/4} = 0.621\text{ m/s}$$

The percent pressure (or head) loss for nozzle meters is given in Fig. 8-59 for $\beta$ = 0.5 to be 62%. Therefore,

$$h_L = (\text{Permanent loss fraction})(\text{Total head loss}) = 0.62(0.32\text{ m}) = \mathbf{0.20\ m\ H_2O}$$

The head loss between the two measurement sections can be determined from the energy equation, which simplifies to ($z_1$ =$z_2$)

$$h_L = \frac{P_1-P_2}{\rho_f g} - \frac{V_2^2-V_1^2}{2g} = h_w - \frac{\left[(D/d)^4-1\right]V_1^2}{2g} = 0.32\text{ m} - \frac{\left[(4/2)^4-1\right](0.621\text{ m/s})^2}{2\left(9.81\text{ m/s}^2\right)} = 0.025\text{ m H}_2\text{O}$$

***Discussion*** The Reynolds number of flow through the pipe is

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(998\text{ kg/m}^3)(0.621\text{ m/s})(0.04\text{ m})}{1.002\times10^{-3}\text{ kg/m}\cdot\text{s}} = 2.47\times10^4$$

Substituting the β and Re values into the orifice discharge coefficient relation gives

$$C_d = 0.9975 - \frac{6.53\beta^{0.5}}{\text{Re}^{0.5}} = 0.9975 - \frac{6.53(0.50)^{0.5}}{(2.47\times10^4)^{0.5}} = 0.968$$

which is almost identical to the assumed value of 0.96.

**8-111E**

**Solution** A Venturi meter equipped with a differential pressure meter is used to measure to flow rate of refrigerant-134a through a horizontal pipe. For a measured pressure drop, the volume flow rate is to be determined.

***Assumptions*** The flow is steady and incompressible.

***Properties*** The density of R-134a is given to be $\rho$ = 83.31 lbm/ft$^3$. The discharge coefficient of Venturi meter is given to be $C_d$ = 0.98.

5 in

2 in

$\Delta P$ = 7.4 psi

Differential pressure meter

***Analysis*** The diameter ratio and the throat area of the meter are

$$\beta = d/D = 2/5 = 0.40$$

$$A_0 = \pi d^2/4 = \pi(2/12\text{ ft})^2/4 = 0.02182\text{ ft}^2$$

Noting that $\Delta P$ = 7.4 psi = 7.4×144 lbf/ft$^2$, the flow rate becomes

$$\dot{V} = A_o C_d \sqrt{\frac{2(P_1 - P_2)}{\rho(1-\beta^4)}}$$

$$= (0.02182\text{ ft}^2)(0.98)\sqrt{\frac{2\times 7.4\times 144\text{ lbf/ft}^2}{(83.31\text{ lbm/ft}^3)((1-0.40^4)}\left(\frac{32.2\text{ lbm}\cdot\text{ft/s}^2}{1\text{ lbf}}\right)}$$

$$= \mathbf{0.622\ ft^3/s}$$

Also, the average flow velocity in the pipe is determined by dividing the flow rate by the cross-sectional area of the pipe,

$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.622\text{ ft}^3/\text{s}}{\pi(5/12\text{ ft})^2/4} = \mathbf{4.56\ ft/s}$$

***Discussion*** Note that the flow rate is proportional to the square root of pressure difference across the Venturi meter.

## Review Problems

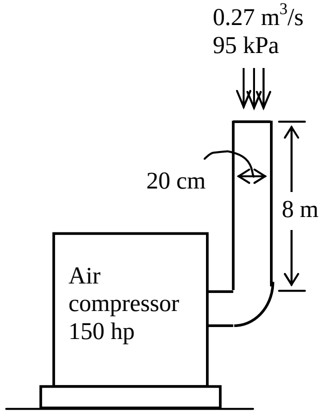
**8-112**

**Solution** A compressor takes in air at a specified rate at the outdoor conditions. The useful power used by the compressor to overcome the frictional losses in the duct is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** Air is an ideal gas. **4** The duct involves no components such as bends, valves, and connectors, and thus minor losses are negligible. **5** The flow section involves no work devices such as fans or turbines.

***Properties*** The properties of air at 1 atm = 101.3 kPa and 15°C are $\rho_0$ = 1.225 kg/m$^3$ and $\mu$ = 1.802×10$^{-5}$ kg/m·s. The roughness of galvanized iron surfaces is $\varepsilon$ = 0.00015 m. The dynamic viscosity is independent of pressure, but density of an ideal gas is proportional to pressure. The density of air at 95 kPa is

$$\rho = (P/P_0)\rho_0 = (95/101.3)(1.225\ \text{kg/m}^3) = 1.149\ \text{kg/m}^3.$$




***Analysis*** The average velocity and the Reynolds number are

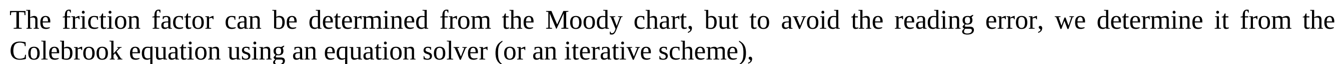
$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} = \frac{0.27\ \text{m}^3/\text{s}}{\pi(0.20\ \text{m})^2/4} = 8.594\ \text{m/s}$$

$$\text{Re} = \frac{\rho V D_h}{\mu} = \frac{(1.149\ \text{kg/m}^3)(8.594\ \text{m/s})(0.20\ \text{m})}{1.802\times10^{-5}\ \text{kg/m}\cdot\text{s}} = 1.096\times10^5$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D = \frac{1.5\times10^{-4}\ \text{m}}{0.20\ \text{m}} = 7.5\times10^{-4}$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),



$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D_h}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{7.5\times10^{-4}}{3.7} + \frac{2.51}{1.096\times10^5\sqrt{f}}\right)$$

It gives $f$ = 0.02109. Then the pressure drop in the duct and the required pumping power become

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.02109\frac{8\ \text{m}}{0.20\ \text{m}}\frac{(1.149\ \text{kg/m}^3)(8.594\ \text{m/s})^2}{2}\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)\left(\frac{1\ \text{Pa}}{1\ \text{N/m}^2}\right) = 35.8\ \text{Pa}$$

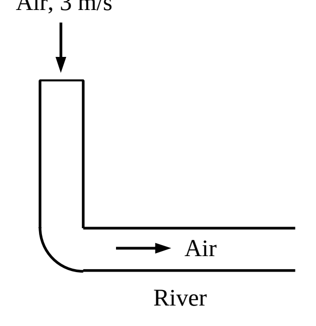
$$\dot{W}_{\text{pump,u}} = \dot{V}\Delta P = (0.27\ \text{m}^3/\text{s})(35.8\ \text{Pa})\left(\frac{1\ \text{W}}{1\ \text{Pa}\cdot\text{m}^3/\text{s}}\right) = \mathbf{9.66\ W}$$

***Discussion*** Note hat the pressure drop in the duct and the power needed to overcome it is very small (relative to 150 hp), and can be disregarded. The friction factor could also be determined easily from the explicit Haaland relation. It would give $f$ = 0.02086, which is very close to the Colebrook value. Also, the power input determined is the mechanical power that needs to be imparted to the fluid. The shaft power will be more than this due to fan inefficiency; the electrical power input will be even more due to motor inefficiency (but probably no more than 20 W).

**8-113**

**Solution** Air enters the underwater section of a circular duct. The fan power needed to overcome the flow resistance in this section of the duct is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** Air is an ideal gas. **4** The duct involves no components such as bends, valves, and connectors. **5** The flow section involves no work devices such as fans or turbines. **6** The pressure of air is 1 atm.




***Properties*** The properties of air at 1 atm and 15°C are $\rho_0$ = 1.225 kg/m$^3$ and $\mu$ = 1.802×10$^{-5}$ kg/m·s. The roughness of stainless steel pipes is $\varepsilon$ = 0.000005 m.

***Analysis*** The volume flow rate and the Reynolds number are

$$\dot{V} = VA_c = V(\pi D^2/4) = (3\text{ m/s})[\pi(0.20\text{ m})^2/4] = 0.0942\text{ m}^3/\text{s}$$

$$\text{Re} = \frac{\rho V D_h}{\mu} = \frac{(1.225\text{ kg/m}^3)(3\text{ m/s})(0.20\text{ m})}{1.802\times10^{-5}\text{ kg/m}\cdot\text{s}} = 4.079\times10^4$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D = \frac{5\times10^{-6}\text{ m}}{0.20\text{ m}} = 2.5\times10^{-5}$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D_h}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{2.5\times10^{-5}}{3.7} + \frac{2.51}{4.079\times10^4\sqrt{f}}\right)$$

It gives $f$ = 0.02195. Then the pressure drop in the duct and the required pumping power become

$$\Delta P = \Delta P_L = f\frac{L}{D}\frac{\rho V^2}{2} = 0.02195\frac{15\text{ m}}{0.2\text{ m}}\frac{(1.225\text{ kg/m}^3)(3\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ Pa}}{1\text{ N/m}^2}\right) = 9.07\text{ Pa}$$

$$\dot{W}_{\text{electric}} = \frac{\dot{W}_{\text{pump, u}}}{\eta_{\text{pump-motor}}} = \frac{\dot{V}\Delta P}{\eta_{\text{pump-motor}}} = \frac{(0.0942\text{ m}^3/\text{s})(9.07\text{ Pa})}{0.62} = \left(\frac{1\text{ W}}{1\text{ Pa}\cdot\text{m}^3/\text{s}}\right) = \mathbf{1.4\text{ W}}$$

***Discussion*** The friction factor could also be determined easily from the explicit Haaland relation. It would give $f$ = 0.02175, which is sufficiently close to 0.02195. Assuming the pipe to be smooth would give 0.02187 for the friction factor, which is almost identical to the $f$ value obtained from the Colebrook relation. Therefore, the duct can be treated as being smooth with negligible error.

---

**8-114**

**Solution** The velocity profile in fully developed laminar flow in a circular pipe is given. The radius of the pipe, the average velocity, and the maximum velocity are to be determined.

***Assumptions*** The flow is steady, laminar, and fully developed.

***Analysis*** The velocity profile in fully developed laminar flow in a circular pipe is

$$u(r) = u_{\max}\left(1-\frac{r^2}{R^2}\right)$$

$u(r)$=$u_{max}$(1-$r^2$/$R^2$)

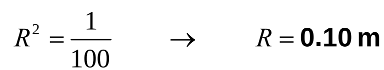
R

r

0

$u_{max}$

The velocity profile in this case is given by $u(r) = 6(1-0.01r^2)$. Comparing the two relations above gives the pipe radius, the maximum velocity, and the average velocity to be



$$R^2 = \frac{1}{100} \quad\rightarrow\quad R = \mathbf{0.10\text{ m}}$$

$$u_{\max} = \mathbf{6\text{ m/s}} \quad\rightarrow\quad V_{avg} = \frac{u_{\max}}{2} = \frac{6\text{ m/s}}{2} = \mathbf{3\text{ m/s}}$$

***Discussion*** In fully developed laminar pipe flow, average velocity is exactly half of maximum (centerline) velocity.

---

**8-115E**

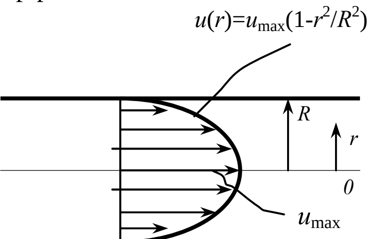
**Solution** The velocity profile in fully developed laminar flow in a circular pipe is given. The volume flow rate, the pressure drop, and the useful pumping power required to overcome this pressure drop are to be determined.

***Assumptions*** **1** The flow is steady, laminar, and fully developed. **2** The pipe is horizontal.

***Properties*** The density and dynamic viscosity of water at 40°F are $\rho$ = 62.42 lbm/ft$^3$ and $\mu$ = 3.74 lbm/ft·h = 1.039×10$^{-3}$ lbm/ft·s, respectively.

***Analysis*** The velocity profile in fully developed laminar flow in a circular pipe is

$$u(r) = u_{max}\left(1 - \frac{r^2}{R^2}\right)$$




The velocity profile in this case is given by

$$u(r) = 0.8(1 - 625r^2)$$

Comparing the two relations above gives the pipe radius, the maximum velocity, and the average velocity to be

$$R^2 = \frac{1}{625} \quad \rightarrow \quad R = 0.04 \text{ ft}$$

$u_{max}$ = 0.8 ft/s

$$V = V_{avg} = \frac{u_{max}}{2} = \frac{0.8 \text{ ft/s}}{2} = 0.4 \text{ ft/s}$$

Then the volume flow rate and the pressure drop become

$$\dot{V} = VA_c = V(\pi R^2) = (0.4 \text{ ft/s})[\pi(0.04 \text{ ft})^2] = \mathbf{0.00201 \text{ ft}^3/s}$$

$$\dot{V}_{horiz} = \frac{\Delta P \pi D^4}{128\mu L} \quad \rightarrow \quad 0.00201 \text{ ft}^3/\text{s} = \frac{(\Delta P)\pi(0.08 \text{ ft})^4}{128(1.039\times10^{-3} \text{ lbm/ft}\cdot\text{s})(80 \text{ ft})}\left(\frac{32.2 \text{ lbm}\cdot\text{ft/s}^2}{1 \text{ lbf}}\right)$$

It gives

$$\Delta P = 5.16 \text{ lbf/ft}^2 = \mathbf{0.0358 \text{ psi}}$$

Then the useful pumping power requirement becomes

$$\dot{W}_{pump,u} = \dot{V}\Delta P = (0.00201 \text{ ft}^3/\text{s})(5.16 \text{ lbf/ft}^2)\left(\frac{1 \text{ W}}{0.737 \text{ lbf}\cdot\text{ft/s}}\right) = \mathbf{0.014 \text{ W}}$$

**Checking** The flow was assumed to be laminar. To verify this assumption, we determine the Reynolds number:

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(62.42 \text{ lbm/ft}^3)(0.4 \text{ ft/s})(0.08 \text{ ft})}{1.039\times10^{-3} \text{ lbm/ft}\cdot\text{s}} = 1922$$

which is less than 2300. Therefore, the flow is laminar.

***Discussion*** Note that the pressure drop across the water pipe and the required power input to maintain flow is negligible. This is due to the very low flow velocity. Such water flows are the exception in practice rather than the rule.

**8-116E**

**Solution** The velocity profile in fully developed laminar flow in a circular pipe is given. The volume flow rate, the pressure drop, and the useful pumping power required to overcome this pressure drop are to be determined.

***Assumptions*** The flow is steady, laminar, and fully developed.

***Properties*** The density and dynamic viscosity of water at 40°F are $\rho$ = 62.42 lbm/ft$^3$ and $\mu$ = 3.74 lbm/ft·h = 1.039×10$^{-3}$ lbm/ft·s, respectively.

***Analysis*** The velocity profile in fully developed laminar flow in a circular pipe is

$$u(r) = u_{\max}\left(1 - \frac{r^2}{R^2}\right)$$

The velocity profile in this case is given by

$$u(r) = 0.8(1 - 625r^2)$$

Comparing the two relations above gives the pipe radius, the maximum velocity, the average velocity, and the volume flow rate to be

$$R^2 = \frac{1}{625} \quad \rightarrow \quad R = 0.04\ \text{ft}$$

$$u_{\max} = 0.8\ \text{ft/s}$$

$$V = V_{avg} = \frac{u_{\max}}{2} = \frac{0.8\ \text{ft/s}}{2} = 0.4\ \text{ft/s}$$

$$\dot{V} = VA_c = V(\pi R^2) = (0.4\ \text{ft/s})[\pi(0.04\ \text{ft})^2] = \mathbf{0.00201\ ft^3/s}$$

For uphill flow with an inclination of 12°, we have θ = +12°, and

$$\rho g L \sin\theta = (62.42\ \text{lbm/ft}^3)(32.2\ \text{ft/s}^2)(80\ \text{ft})\sin 12°\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right) = 1038\ \text{lbf/ft}^2$$

$$\dot{V}_{\text{uphill}} = \frac{(\Delta P - \rho g L \sin\theta)\pi D^4}{128\mu L} \rightarrow 0.00201\ \text{ft}^3\text{/s} = \frac{(\Delta P - 1038)\pi(0.08\ \text{ft})^4}{128(1.039\times 10^{-3}\ \text{lbm/ft}\cdot\text{s})(80\ \text{ft})}\left(\frac{32.2\ \text{lbm}\cdot\text{ft/s}^2}{1\ \text{lbf}}\right)$$

It gives

$$\Delta P = 1043\ \text{lbf/ft}^2 = 7.24\ \text{psi}$$

Then the useful pumping power requirement becomes

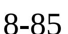
$$\dot{W}_{\text{pump, u}} = \dot{V}\Delta P = (0.00201\ \text{ft}^3\text{/s})(1043\ \text{lbf/ft}^2)\left(\frac{1\ \text{W}}{0.737\ \text{lbf}\cdot\text{ft/s}}\right) = \mathbf{2.84\ W}$$

**Checking** The flow was assumed to be laminar. To verify this assumption, we determine the Reynolds number:

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{(62.42\ \text{lbm/ft}^3)(0.4\ \text{ft/s})(0.08\ \text{ft})}{1.039\times 10^{-3}\ \text{lbm/ft}\cdot\text{s}} = 1922$$

which is less than 2300. Therefore, the flow is laminar.

***Discussion*** Note that the pressure drop across the water pipe and the required power input to maintain flow is negligible. This is due to the very low flow velocity. Such water flows are the exception in practice rather than the rule.



**8-117**

**Solution** Water is discharged from a water reservoir through a circular pipe of diameter $D$ at the side wall at a vertical distance $H$ from the free surface with a reentrant section. A relation for the "equivalent diameter" of the reentrant pipe for use in relations for frictionless flow through a hole is to be obtained.

***Assumptions*** **1** The flow is steady and incompressible. **2** The reservoir is open to the atmosphere so that the pressure is atmospheric pressure at the free surface. **3** The water level in the reservoir remains constant. **4** The pipe is horizontal. **5** The entrance effects are negligible, and thus the flow is fully developed and the friction factor $f$ is constant. **6** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Properties*** The loss coefficient is $K_L = 0.8$ for the reentrant section, and $K_L = 0$ for the "frictionless" flow.

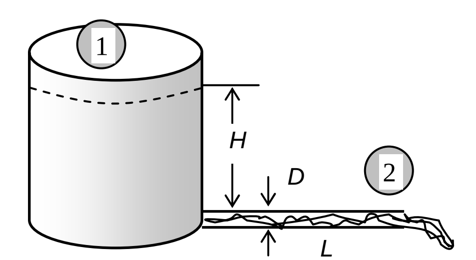
***Analysis*** We take point 1 at the free surface of the reservoir and point 2 at the exit of the pipe, which is also taken as the reference level ($z_2 = 0$). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface of the reservoir is zero ($V_1 = 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad H=\alpha_2\frac{V_2^2}{2g}+h_L$$

where $\alpha_2 = 1$ and

$$h_L=h_{L,\text{total}}=h_{L,\text{major}}+h_{L,\text{minor}}=\left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}=\left(f\frac{L}{D}+K_L\right)\frac{V_2^2}{2g}$$

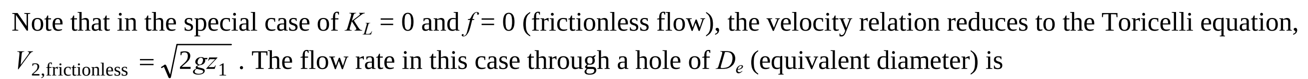
since the diameter of the pipe is constant. Substituting and solving for $V_2$ gives




$$H=\frac{V_2^2}{2g}+\left(f\frac{L}{D}+K_L\right)\frac{V_2^2}{2g} \quad\rightarrow\quad V_2=\sqrt{\frac{2gH}{1+fL/D+K_L}}$$

Then the volume flow rate becomes

$$\dot{V}=A_cV_2=\frac{\pi D^2}{4}\sqrt{\frac{2gH}{1+fL/D+K_L}} \qquad (1)$$



Note that in the special case of $K_L = 0$ and $f = 0$ (frictionless flow), the velocity relation reduces to the Toricelli equation, $V_{2,\text{frictionless}}=\sqrt{2gz_1}$. The flow rate in this case through a hole of $D_e$ (equivalent diameter) is

$$\dot{V}=A_{c,\text{equiv}}V_{2,\text{frictionless}}=\frac{\pi D_{\text{equiv}}^2}{4}\sqrt{2gH} \qquad (2)$$

Setting Eqs. (1) and (2) equal to each other gives the desired relation for the equivalent diameter,

$$\frac{\pi D_{\text{equiv}}^2}{4}\sqrt{2gH}=\frac{\pi D^2}{4}\sqrt{\frac{2gH}{1+fL/D+K_L}}$$

which gives

$$D_{\text{equiv}}=\frac{D}{(1+fL/D+K_L)^{1/4}}=\frac{D}{(1+0.018\times 10/0.04+0.8)^{1/4}}=\mathbf{0.63D=0.025\ \text{m}}$$

***Discussion*** Note that the effect of frictional losses of a pipe with a reentrant section is to reduce the diameter by about 40% in this case. Also, noting that the flow rate is proportional to the square of the diameter, we have $\dot{V}\propto D_{\text{equiv}}^2=(0.63D)^2=0.40D^2$. Therefore, the flow rate through a sharp-edged entrance is about two-thirds less compared to the frictionless flow case.

**8-118**

**Solution** A water tank open to the atmosphere is initially filled with water. The tank is drained to the atmosphere through a 90° horizontal bend of negligible length. The flow rate is to be determined for the cases of the bend being a flanged smooth bend and a miter bend without vanes.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is turbulent so that the tabulated value of the loss coefficient can be used. **3** The water level in the tank remains constant. **4** The length of the bend and thus the frictional loss associated with its length is negligible. **5** The entrance is well-rounded, and the entrance loss is negligible.

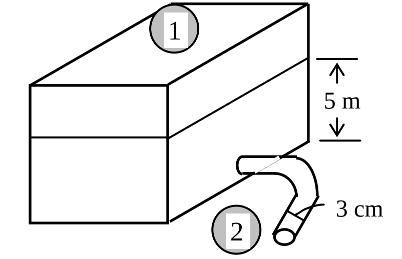
***Properties*** The loss coefficient is $K_L = 0.3$ for a flanged smooth bend and $K_L = 1.1$ for a miter bend without vanes.

***Analysis*** (*a*) We take point 1 at the free surface of the tank, and point 2 at the exit of the bend, which is also taken as the reference level ($z_2 = 0$). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad z_1 = \alpha_2 \frac{V_2^2}{2g} + h_L$$

where the head loss is expressed as $h_L = K_L \frac{V^2}{2g}$. Substituting and solving for $V_2$ gives

$$z_1 = \alpha_2 \frac{V_2^2}{2g} + K_L \frac{V_2^2}{2g} \quad \rightarrow \quad 2gz_1 = V_2^2(\alpha_2 + K_L) \quad \rightarrow \quad V_2 = \sqrt{\frac{2gz_1}{\alpha_2 + K_L}}$$




Then the flow rate becomes

$$\dot{V} = A_{\text{pipe}} V_2 = \frac{\pi D^2}{4} \sqrt{\frac{2gz_1}{\alpha_2 + K_L}}$$

(*a*) Case 1 **Flanged smooth bend ($K_L$ = 0.3):**

$$\dot{V} = A_c V_2 = \frac{\pi D^2}{4} \sqrt{\frac{2gz_1}{\alpha_2 + K_L}} = \frac{\pi (0.03\text{ m})^2}{4} \sqrt{\frac{2(9.81\text{ m/s}^2)(5\text{ m})}{1.05 + 0.3}} = \mathbf{0.00603\ m^3/s = 6.03\ L/s}$$

(*b*) Case 2 **Miter bend without vanes ($K_L$ = 1.1):**

$$\dot{V} = A_c V_2 = \frac{\pi D^2}{4} \sqrt{\frac{2gz_1}{\alpha_2 + K_L}} = \frac{\pi (0.03\text{ m})^2}{4} \sqrt{\frac{2(9.81\text{ m/s}^2)(5\text{ m})}{1.05 + 1.1}} = \mathbf{0.00478\ m^3/s = 4.78\ L/s}$$

***Discussion*** Note that the type of bend used has a significant effect on the flow rate, and a conscious effort should be made when selecting components in a piping system. If the effect of the kinetic energy correction factor is neglected, $\alpha_2 = 1$ and the flow rates become

**(*a*) Case 1 ($K_L$ = 0.3):** $$\dot{V} = A_c V_2 = \frac{\pi D^2}{4} \sqrt{\frac{2gz_1}{1 + K_L}} = \frac{\pi (0.03\text{ m})^2}{4} \sqrt{\frac{2(9.81\text{ m/s}^2)(5\text{ m})}{1 + 0.3}} = 0.00614\ \text{m}^3/\text{s}$$

**(*b*) Case 2 ($K_L$ = 1.1):** $$\dot{V} = A_c V_2 = \frac{\pi D^2}{4} \sqrt{\frac{2gz_1}{1 + K_L}} = \frac{\pi (0.03\text{ m})^2}{4} \sqrt{\frac{2(9.81\text{ m/s}^2)(5\text{ m})}{1 + 1.1}} = 0.00483\ \text{m}^3/\text{s}$$

Therefore, the effect of the kinetic energy correction factor is (6.14 - 6.03)/6.03 = 1.8% and (4.83 - 4.78)/4.83 = 1.0%, which is negligible.

**8-119** **[*Also solved using EES on enclosed DVD*]**

**Solution** The piping system of a geothermal district heating system is being designed. The pipe diameter that will optimize the initial system cost and the energy cost is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The minor losses are negligible because of the large length-to-diameter ratio and the relatively small number of components that cause minor losses, the only significant energy loss arises from pipe friction. **4** The piping system is horizontal. **5** The properties of geothermal water are the same as fresh water. **6** The friction factor is constant at the given value. **7** The interest rate, the inflation rate, and the salvage value of the system are all zero. **8** The flow rate through the system remains constant.

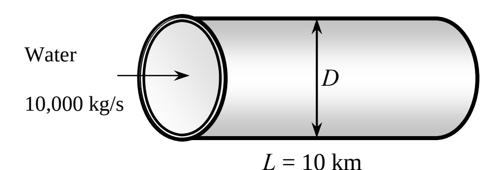
***Properties*** We take the density of water to be $\rho$ = 1000 kg/m$^3$. The friction factor is given to be $f$ = 0.015.

***Analysis*** The system operates in a loop, and thus we can take any point in the system as points 1 and 2 (the same point), and thus $z_1 = z_2$, $V_1 = V_2$, and $P_1 = P_2$. Then the energy equation for this piping system simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_1\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad h_{\text{pump, u}}=h_L$$

That is, the pumping power is to be used to overcome the head losses due to friction in flow. When the minor losses are disregarded, the head loss for fully developed flow in a pipe of length $L$ and diameter $D$ can be expressed as

$$\Delta P = f\frac{L}{D}\frac{\rho V^2}{2}$$




The flow rate of geothermal water is

$$\dot{V}=\frac{\dot{m}}{\rho}=\frac{10{,}000\text{ kg/s}}{1000\text{ kg/m}^3}=10\text{ m}^3\text{/s}$$

To expose the dependence of pressure drop on diameter, we express it in terms of the flow rate as

$$\Delta P = f\frac{L}{D}\frac{\rho V^2}{2}=f\frac{L}{D}\frac{\rho}{2}\left(\frac{\dot{V}}{\pi D^2/4}\right)^2=f\frac{16L}{D}\frac{\rho\dot{V}^2}{2\pi^2D^4}=f\frac{8L}{D^5}\frac{\rho\dot{V}^2}{\pi^2}$$

Then the required pumping power can be expressed as

$$\dot{W}_{\text{pump}}=\frac{\dot{W}_{\text{pump,u}}}{\eta_{\text{pump-motor}}}=\frac{\dot{V}\Delta P}{\eta_{\text{pump-motor}}}=\frac{\dot{V}}{\eta_{\text{pump-motor}}}f\frac{8L}{D^5}\frac{\rho\dot{V}^2}{\pi^2}=f\frac{8L}{D^5}\frac{\rho\dot{V}^3}{\eta_{\text{pump-motor}}\pi^2}$$

Note that the pumping power requirement is proportional to $f$ and $L$, consistent with our intuitive expectation. Perhaps not so obvious is that power is proportional to the *cube* of flow rate. The fact that the *power is inversely proportional to pipe diameter D to the fifth power* averages that a slight increase in pipe diameter will manifest as a tremendous reduction in power dissipation due to friction in a long pipeline. Substituting the given values and expressing the diameter $D$ in meters,

$$\dot{W}_{\text{pump}}=(0.015)\frac{8(10{,}000\text{ m})}{D^5\text{ m}^5}\frac{(1000\text{ kg/m}^3)(10\text{ m}^3\text{/s})^3}{\pi^2(0.80)}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}}\right)=\frac{1.52\times10^5}{D^5}\text{ kW}$$

The number of hours in one year are 24×365 = 8760 h. Then the total amount of electric power used and its cost per year are

$$E_{\text{pump}}=\dot{W}_{\text{pump}}\Delta t=\frac{1.52\times10^5}{D^5}(8760\text{ h})=\frac{1.332\times10^9}{D^5}\text{ kWh/yr}$$

$$\text{Energy cost}=E_{\text{pump}}\times\text{Unit cost}=\left(\frac{1.332\times10^9}{D^5}\text{ kWh/y}\right)(\$0.06\text{/kWh})=\frac{7.99\times10^7}{D^5}\text{ \$/yr}$$

The installation cost of the system with a 30-year lifetime is given to be Cost = \$10$^6$ $D^2$ where $D$ is in meters. The annual cost of the system is then 1/30$^{\text{th}}$ of it, which is

$$\text{System cost}=\frac{\text{Total cost}}{\text{Life time}}=\frac{\$10^6D^2}{30\text{ yr}}=\$3.33\times10^4D^2\text{ (per year)}$$

Then the total annual cost of the system (installation + operating) becomes

$$\text{Total cost = Energy cost + System cost} = \frac{7.99\times 10^{7}}{D^{5}} + 3.33\times 10^{4} D^{2} \quad \$/\text{yr}$$

The optimum pipe diameter is the value that minimizes this total, and it is determined by taking the derivative of the total cost with respect to $D$ and setting it equal to zero,

$$\frac{\partial(\text{Total cost})}{\partial D} = -5\times \frac{7.99\times 10^{7}}{D^{6}} + 2\times 3.33\times 10^{4} D = 0$$

Simplifying gives $D^7$ = 5998 whose solution is

$D$ = **3.5 m**

**This is the optimum pipe diameter that minimizes the total cost of the system under stated assumptions**. A larger diameter pipe will increase the system cost more than it decreases the energy cost, and a smaller diameter pipe will increase the system cost more than it decreases the energy cost.

***Discussion*** The assumptions of zero interest and zero inflation are not realistic, and an actual economic analysis must consider these factors as they have a major effect on the pipe diameter. This is done by considering the time value of money, and expressing all the costs at the same time. Pipe purchase is a present cost, and energy expenditures are future annual costs spread out over the project lifetime. Thus, to provide consistent dollar comparisons between initial and future costs, all future energy costs must be expressed as a single present lump sum to reflect the time-value of money. Then we can compare pipe and energy costs on a consistent basis. Economists call the necessary factor the "Annuity Present Value Factor", F. If interest rate is 10% per year with n = 30 years, then F = 9.427. Thus, if power costs $1,000,000/year for the next 30 years, then the present value of those future payments is $9,427,000 (and not $30,000,000!) if money is worth 10%. Alternatively, if you must pay $1,000,000 every year for 30 years, and you can today invest $9,437,000 at 10%, then you can meet 30 years of payments at the end of each year. The energy cost in this case can be determined by dividing the energy cost expression developed above by 9.427. This will result in a pipe diameter of $D$ = 2.5 m. In an actual design, we also need to calculate the average flow velocity and the pressure head to make sure that they are reasonable. For a pipe diameter of 2.5 m, for example, the average flow velocity is 1.47 m/s and the pump pressure head is 5.6 m.

**8-120**

**Solution** Water is drained from a large reservoir through two pipes connected in series at a specified rate using a pump. The required pumping head and the minimum pumping power are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The pipes are horizontal. **3** The entrance effects are negligible, and thus the flow is fully developed. **4** The flow is turbulent so that the tabulated value of the loss coefficients can be used. **5** The pipes involve no components such as bends, valves, and other connectors that cause additional minor losses. **6** The reservoir is open to the atmosphere so that the pressure is atmospheric pressure at the free surface. **7** The water level in the reservoir remains constant. **8** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Properties*** The density and dynamic viscosity of water at 15°C are $\rho$ = 999.1 kg/m$^3$ and $\mu$ = 1.138×10$^{-3}$ kg/m·s. The loss coefficient is $K_L$ = 0.5 for a sharp-edged entrance. The roughness of cast iron pipes is $\varepsilon$ = 0.00026 m.

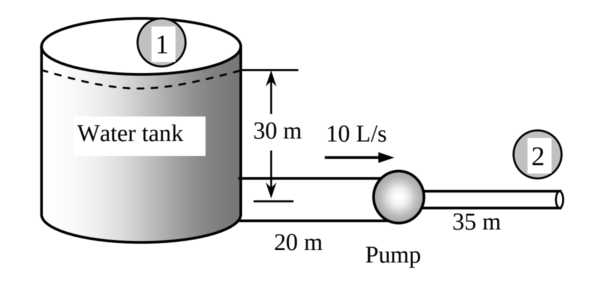
***Analysis*** We take point 1 at the free surface of the tank, and point 2 and the reference level at the centerline of the pipe ($z_2$ = 0). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface of the tank is very low ($V_1 \cong 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad z_1+h_{\text{pump, u}}=\alpha_2\frac{V_2^2}{2g}+h_L$$

where $\alpha_2$ = 1 and

$$h_L=h_{L,\text{total}}=h_{L,\text{major}}+h_{L,\text{minor}}=\sum\left(f\frac{L}{D}+\sum K_L\right)\frac{V^2}{2g}$$

and the summation is over two pipes. Noting that the two pipes are connected in series and thus the flow rate through each of them is the same, the head loss for each pipe is determined as follows (we designate the first pipe by 1 and the second one by 2):




**Pipe 1**: $V_1=\frac{\dot V}{A_{c1}}=\frac{\dot V}{\pi D_1^2/4}=\frac{0.018\text{ m}^3/\text{s}}{\pi(0.06\text{ m})^2/4}=6.366\text{ m/s}$

$$\text{Re}_1=\frac{\rho V_1 D_1}{\mu}=\frac{(999.1\text{ kg/m}^3)(6.366\text{ m/s})(0.06\text{ m})}{1.138\times10^{-3}\text{ kg/m}\cdot\text{s}}=335{,}300$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D_1=\frac{0.00026\text{ m}}{0.06\text{ m}}=0.00433$$

The friction factor corresponding to this relative roughness and the Reynolds number is, from the Colebrook equation,

$$\frac{1}{\sqrt{f_1}}=-2.0\log\left(\frac{\varepsilon/D_1}{3.7}+\frac{2.51}{\text{Re}_1\sqrt{f_1}}\right)\rightarrow\frac{1}{\sqrt{f_1}}=-2.0\log\left(\frac{0.00433}{3.7}+\frac{2.51}{335{,}300\sqrt{f_1}}\right)$$

It gives $f_1$ = 0.02941. The only minor loss is the entrance loss, which is $K_L$ = 0.5. Then the total head loss of the first pipe becomes

$$h_{L1}=\left(f_1\frac{L_1}{D_1}+\sum K_L\right)\frac{V_1^2}{2g}=\left((0.02941)\frac{20\text{ m}}{0.06\text{ m}}+0.5\right)\frac{(6.366\text{ m/s})^2}{2(9.81\text{ m/s}^2)}=21.3\text{ m}$$

**Pipe 2**: $V_2=\frac{\dot V}{A_{c2}}=\frac{\dot V}{\pi D_2^2/4}=\frac{0.018\text{m}^3/\text{s}}{\pi(0.04\text{ m})^2/4}=14.32\text{ m/s}$

$$\text{Re}_2=\frac{\rho V_2 D_2}{\mu}=\frac{(999.1\text{ kg/m}^3)(14.32\text{ m/s})(0.04\text{ m})}{1.138\times10^{-3}\text{ kg/m}\cdot\text{s}}=502{,}900$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D_2=\frac{0.00026\text{ m}}{0.04\text{ m}}=0.0065$$

The friction factor corresponding to this relative roughness and the Reynolds number is, from the Colebrook equation,

$$\frac{1}{\sqrt{f_2}} = -2.0\log\left(\frac{\varepsilon / D_2}{3.7} + \frac{2.51}{\text{Re}_2\sqrt{f_2}}\right) \rightarrow \frac{1}{\sqrt{f_2}} = -2.0\log\left(\frac{0.0065}{3.7} + \frac{2.51}{502{,}900\sqrt{f_2}}\right)$$

It gives $f_2$ = 0.03309. The second pipe involves no minor losses. Note that we do not consider the exit loss unless the exit velocity is dissipated within the system considered (in this case it is not). Then the head loss for the second pipe becomes

$$h_{L2} = f_2\frac{L_2}{D_2}\frac{V_2^2}{2g} = (0.03309)\frac{35\text{ m}}{0.04\text{ m}}\frac{(14.32\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 302.6\text{ m}$$

The total head loss for two pipes connected in series is the sum of the head losses of the two pipes,

$$h_L = h_{L,\text{total}} = h_{L1} + h_{L2} = 21.3 + 302.6 = 323.9\text{ m}$$

Then the pumping head and the minimum pumping power required (the pumping power in the absence of any inefficiencies of the pump and the motor, which is equivalent to the useful pumping power) become

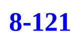
$$h_{\text{pump, u}} = \alpha_2\frac{V_2^2}{2g} + h_L - z_1 = (1)\frac{(14.32\text{ m/s})^2}{2(9.81\text{ m/s}^2)} + 323.9 - 30 = 304.4\text{ m}$$

$$\dot{W}_{\text{pump, u}} = \dot{V}\Delta P = \rho\dot{V}gh_{\text{pump, u}}$$

$$= (999.1\text{ kg/m}^3)(0.018\text{ m}^3\text{/s})(9.81\text{ m/s}^2)(304.4\text{ m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}}\right) = \mathbf{53.7\text{ kW}}$$

Therefore, the pump must supply a minimum of 53.7 kW of useful mechanical energy to water.

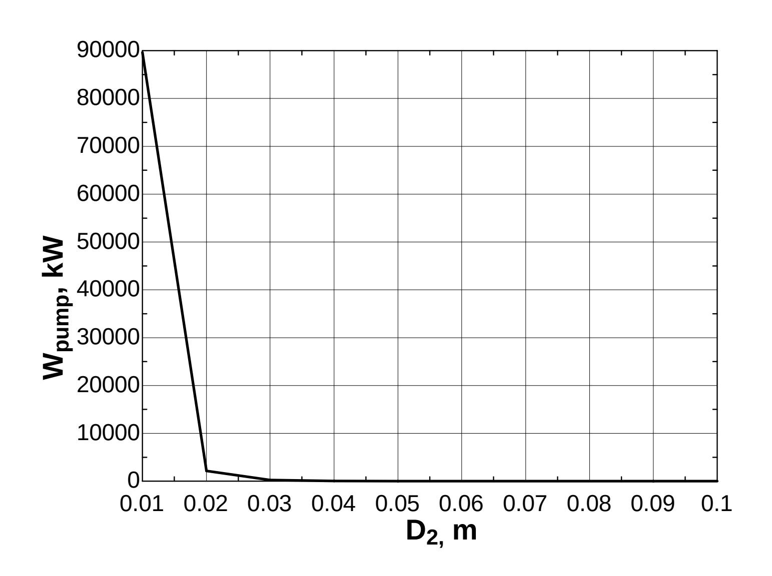
***Discussion*** Note that the shaft power of the pump must be greater than this to account for the pump inefficiency, and the electrical power supplied must be even greater to account for the motor inefficiency.


**8-121** 

**Solution** In the previous problem, the effect of second pipe diameter on required pumping head for the same flow rate is to be investigated by varying the pipe diameter from 1 cm to 10 cm in increments of 1 cm.

***Analysis*** The EES *Equations* window is printed below, followed by the tabulated and plotted results.

```
rho=999.1
mu=0.001138
nu=mu/rho
Vdot=0.018 "m3/s"
g=9.81 "m/s2"
z1=30 "m"
L1=20 "m"
D1=0.06 "m"
Ac1=pi*D1^2/4
Re1=V1*D1/nu
V1=Vdot/Ac1
eps1=0.00026
rf1=eps1/D1
1/sqrt(f1)=-2*log10(rf1/3.7+2.51/(Re1*sqrt(f1)))
KL1=0.5
HL1=(f1*L1/D1+KL1)*V1^2/(2*g)
L2=35
Re2=V2*D2/nu
V2=Vdot/(pi*D2^2/4)
eps2=0.00026
rf2=eps2/D2
1/sqrt(f2)=-2*log10(rf2/3.7+2.51/(Re2*sqrt(f2)))
HL2=f2*(L2/D2)*V2^2/(2*g)
HL=HL1+HL2
hpump=V2^2/(2*g)+HL-z1
Wpump=rho*Vdot*g*hpump/1000 "kW"
```




| $D_2$, m | $W_{pump}$, kW | $h_{L2}$, m | Re |
|---|---|---|---|
| 0.01 | 89632.5 | 505391.6 | 2.012E+06 |
| 0.02 | 2174.7 | 12168.0 | 1.006E+06 |
| 0.03 | 250.8 | 1397.1 | 6.707E+05 |
| 0.04 | 53.7 | 302.8 | 5.030E+05 |
| 0.05 | 15.6 | 92.8 | 4.024E+05 |
| 0.06 | 5.1 | 35.4 | 3.353E+05 |
| 0.07 | 1.4 | 15.7 | 2.874E+05 |
| 0.08 | -0.0 | 7.8 | 2.515E+05 |
| 0.09 | -0.7 | 4.2 | 2.236E+05 |
| 0.10 | -1.1 | 2.4 | 2.012E+05 |

***Discussion*** Clearly, the power decreases quite rapidly with increasing diameter. This is not surprising, since the irreversible frictional head loss (major head loss) decreases significantly with increasing pipe diameter.

**8-122**

**Solution** Two pipes of identical diameter and material are connected in parallel. The length of one of the pipes is twice the length of the other. The ratio of the flow rates in the two pipes is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is fully turbulent in both pipes and thus the friction factor is independent of the Reynolds number (it is the same for both pipes since they have the same material and diameter). **3** The minor losses are negligible.

***Analysis*** When two pipes are parallel in a piping system, the head loss for each pipe must be same. When the minor losses are disregarded, the head loss for fully developed flow in a pipe of length *L* and diameter *D* can be expressed as

$$h_L = f\frac{L}{D}\frac{V^2}{2g} = f\frac{L}{D}\frac{1}{2g}\left(\frac{\dot{V}}{A_c}\right)^2 = f\frac{L}{D}\frac{1}{2g}\left(\frac{\dot{V}}{\pi D^2/4}\right)^2 = 8f\frac{L}{D}\frac{1}{g}\frac{\dot{V}^2}{\pi^2 D^4} = 8f\frac{L}{g\pi^2}\frac{\dot{V}^2}{D^5}$$

Solving for the flow rate gives

$$\dot{V} = \sqrt{\frac{\pi^2 h_L g D^5}{8fL}} = \frac{k}{\sqrt{L}} \qquad (k \text{ is a constant})$$

When the pipe diameter, friction factor, and the head loss is constant, which is the case here for parallel connection, the flow rate becomes inversely proportional to the square root of length *L*. Therefore, when the length is doubled, the flow rate will decrease by a factor of $2^{0.5} = 1.41$ since

If $$\dot{V}_A = \frac{k}{\sqrt{L_A}}$$

Then $$\dot{V}_B = \frac{k}{\sqrt{L_B}} = \frac{k}{\sqrt{2L_A}} = \frac{k}{\sqrt{2}\sqrt{L_A}} = \frac{\dot{V}_A}{\sqrt{2}} = 0.707\dot{V}_A$$



Therefore, the ratio of the flow rates in the two pipes is **0.707**.

***Discussion*** Even though one pipe is twice as long as the other, the volume flow rate in the shorter pipe is *not* twice as much – the relationship is nonlinear.

**8-123** **[*Also solved using EES on enclosed DVD*]**

**Solution** A pipeline that transports oil at a specified rate branches out into two parallel pipes made of commercial steel that reconnects downstream. The flow rates through each of the parallel pipes are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** Entrance effects are negligible, and thus the flow is fully developed. **3** Minor losses are disregarded. **4** Flows through both pipes are turbulent (to be verified).

***Properties*** The density and dynamic viscosity of oil at 40°C are $\rho$ = 876 kg/m$^3$ and $\mu$ = 0.2177 kg/m⋅s. The roughness of commercial steel pipes is $\varepsilon$ = 0.000045 m.

***Analysis*** This problem cannot be solved directly since the velocities (or flow rates) in the pipes are not known. Below we will set up the equations to be solved by an equation solver. The head loss in two parallel branches must be the same, and the total flow rate must the sum of the flow rates in the parallel branches. Therefore,

$$h_{L,1} = h_{L,2} \qquad (1)$$

$$\dot{V} = \dot{V}_1 + \dot{V}_2 \quad \rightarrow \quad \dot{V}_1 + \dot{V}_2 = 3 \qquad (2)$$

We designate the 30-cm diameter pipe by 1 and the 45-cm diameter pipe by 2. The average velocity, the relative roughness, the Reynolds number, friction factor, and the head loss in each pipe are expressed as

$$V_1 = \frac{\dot{V}_1}{A_{c,1}} = \frac{\dot{V}_1}{\pi D_1^2/4} \quad \rightarrow \quad V_1 = \frac{\dot{V}_1}{\pi (0.30\text{m})^2/4} \qquad (3)$$

$$V_2 = \frac{\dot{V}_2}{A_{c,2}} = \frac{\dot{V}_2}{\pi D_2^2/4} \quad \rightarrow \quad V_2 = \frac{\dot{V}_2}{\pi (0.45\text{m})^2/4} \qquad (4)$$

$$\text{rf}_1 = \frac{\varepsilon_1}{D_1} = \frac{4.5\times10^{-5}\text{ m}}{0.30\text{ m}} = 1.5\times10^{-4}$$

$$\text{rf}_2 = \frac{\varepsilon_2}{D_2} = \frac{4.5\times10^{-5}\text{ m}}{0.45\text{ m}} = 1\times10^{-4}$$

$$\text{Re}_1 = \frac{\rho V_1 D_1}{\mu} \quad \rightarrow \quad \text{Re}_1 = \frac{(876\text{ kg/m}^3)\mathbf{V}_1(0.30\text{ m})}{0.2177\text{ kg/m}\cdot s} \qquad (5)$$

$$\text{Re}_2 = \frac{\rho V_2 D_2}{\mu} \quad \rightarrow \quad \text{Re}_2 = \frac{(876\text{ kg/m}^3)\mathbf{V}_2(0.45\text{ m})}{0.2177\text{ kg/m}\cdot s} \qquad (6)$$

$$\frac{1}{\sqrt{f_1}} = -2.0\log\left(\frac{\varepsilon/D_1}{3.7} + \frac{2.51}{\text{Re}_1\sqrt{f_1}}\right) \quad \rightarrow \quad \frac{1}{\sqrt{f_1}} = -2.0\log\left(\frac{1.5\times10^{-4}}{3.7} + \frac{2.51}{\text{Re}_1\sqrt{f_1}}\right) \qquad (7)$$

$$\frac{1}{\sqrt{f_2}} = -2.0\log\left(\frac{\varepsilon/D_2}{3.7} + \frac{2.51}{\text{Re}_2\sqrt{f_2}}\right) \quad \rightarrow \quad \frac{1}{\sqrt{f_2}} = -2.0\log\left(\frac{1\times10^{-4}}{3.7} + \frac{2.51}{\text{Re}_2\sqrt{f_2}}\right) \qquad (8)$$

$$h_{L,1} = f_1\frac{L_1}{D_1}\frac{V_1^2}{2g} \quad \rightarrow \quad h_{L,1} = f_1\frac{500\text{ m}}{0.30\text{ m}}\frac{V_1^2}{2(9.81\text{ m/s}^2)} \qquad (9)$$

$$h_{L,2} = f_2\frac{L_2}{D_2}\frac{V_2^2}{2g} \quad \rightarrow \quad h_{L,2} = f_2\frac{800\text{ m}}{0.45\text{ m}}\frac{V_2^2}{2(9.81\text{ m/s}^2)} \qquad (10)$$

This is a system of 10 equations in 10 unknowns, and solving them simultaneously by an equation solver gives

$\dot{V}_1 = \mathbf{0.91\ m^3/s}$, $\dot{V}_2 = \mathbf{2.09\ m^3/s}$,

$V_1$ = 12.9 m/s, $V_2$ = 13.1 m/s, $h_{L,1} = h_{L,2} = 392$ m

$\text{Re}_1$ = 15,540, $\text{Re}_2$ = 23,800, $f_1$ = 0.02785, $f_2$ = 0.02505

Note that Re > 4000 for both pipes, and thus the assumption of turbulent flow is verified.

***Discussion*** This problem can also be solved by using an iterative approach, but it would be very time consuming. Equation solvers such as EES are invaluable for theses kinds of problems.

**8-124**

**Solution** The piping of a district heating system that transports hot water at a specified rate branches out into two parallel pipes made of commercial steel that reconnects downstream. The flow rates through each of the parallel pipes are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed. **3** The minor losses are given to be negligible. **4** Flows through both pipes are turbulent (to be verified).

***Properties*** The density and dynamic viscosity of water at 100°C are $\rho$ = 957.9 kg/m$^3$ and $\mu$ = 0.282×10$^{-3}$ kg/m·s. The roughness of commercial steel pipes is $\varepsilon$ = 0.000045 m.

***Analysis*** This problem cannot be solved directly since the velocities (or flow rates) in the pipes are not known. Below we will set up the equations to be solved by an equation solver. The head loss in two parallel branches must be the same, and the total flow rate must the sum of the flow rates in the parallel branches. Therefore,

$$h_{L,1} = h_{L,2} \qquad (1)$$

$$\dot{V} = \dot{V}_1 + \dot{V}_2 \;\rightarrow\; \dot{V}_1 + \dot{V}_2 = 3 \qquad (2)$$

We designate the 30-cm diameter pipe by 1 and the 45-cm diameter pipe by 2. The average velocity, the relative roughness, the Reynolds number, friction factor, and the head loss in each pipe are expressed as

$$V_1 = \frac{\dot{V}_1}{A_{c,1}} = \frac{\dot{V}_1}{\pi D_1^2/4} \;\rightarrow\; V_1 = \frac{\dot{V}_1}{\pi (0.30\text{m})^2/4} \qquad (3)$$

$$V_2 = \frac{\dot{V}_2}{A_{c,2}} = \frac{\dot{V}_2}{\pi D_2^2/4} \;\rightarrow\; V_2 = \frac{\dot{V}_2}{\pi (0.45\text{m})^2/4} \qquad (4)$$

$$\text{rf}_1 = \frac{\varepsilon_1}{D_1} = \frac{4.5\times10^{-5}\text{ m}}{0.30\text{ m}} = 1.5\times10^{-4}$$

$$\text{rf}_2 = \frac{\varepsilon_2}{D_2} = \frac{4.5\times10^{-5}\text{ m}}{0.45\text{ m}} = 1\times10^{-4}$$

$$\text{Re}_1 = \frac{\rho V_1 D_1}{\mu} \;\rightarrow\; \text{Re}_1 = \frac{(957.9\text{ kg/m}^3)V_1(0.30\text{ m})}{0.282\times10^{-3}\text{ kg/m}\cdot s} \qquad (5)$$

$$\text{Re}_2 = \frac{\rho V_2 D_2}{\mu} \;\rightarrow\; \text{Re}_2 = \frac{(957.9\text{ kg/m}^3)V_2(0.45\text{ m})}{0.282\times10^{-3}\text{ kg/m}\cdot s} \qquad (6)$$

$$\frac{1}{\sqrt{f_1}} = -2.0\log\left(\frac{\varepsilon/D_1}{3.7} + \frac{2.51}{\text{Re}_1\sqrt{f_1}}\right) \;\rightarrow\; \frac{1}{\sqrt{f_1}} = -2.0\log\left(\frac{1.5\times10^{-4}}{3.7} + \frac{2.51}{\text{Re}_1\sqrt{f_1}}\right) \qquad (7)$$

$$\frac{1}{\sqrt{f_2}} = -2.0\log\left(\frac{\varepsilon/D_2}{3.7} + \frac{2.51}{\text{Re}_2\sqrt{f_2}}\right) \;\rightarrow\; \frac{1}{\sqrt{f_2}} = -2.0\log\left(\frac{1\times10^{-4}}{3.7} + \frac{2.51}{\text{Re}_2\sqrt{f_2}}\right) \qquad (8)$$

$$h_{L,1} = f_1\frac{L_1}{D_1}\frac{V_1^2}{2g} \;\rightarrow\; h_{L,1} = f_1\frac{500\text{ m}}{0.30\text{ m}}\frac{V_1^2}{2(9.81\text{ m/s}^2)} \qquad (9)$$

$$h_{L,2} = f_2\frac{L_2}{D_2}\frac{V_2^2}{2g} \;\rightarrow\; h_{L,2} = f_2\frac{800\text{ m}}{0.45\text{ m}}\frac{V_2^2}{2(9.81\text{ m/s}^2)} \qquad (10)$$

This is a system of 10 equations in 10 unknowns, and their simultaneous solution by an equation solver gives

$\dot{V}_1$ = **0.919 m$^3$/s**, $\dot{V}_2$ = **2.08 m$^3$/s**,

$V_1$ = 13.0 m/s, $V_2$ = 13.1 m/s, $h_{L,1} = h_{L,2} = 187$ m

$\text{Re}_1$ = 1.324×10$^7$, $\text{Re}_2$ = 2.00×10$^7$, $f_1$ = 0.0131, $f_2$ = 0.0121

Note that Re > 4000 for both pipes, and thus the assumption of turbulent flow is verified.

***Discussion*** This problem can also be solved by using a trial-and-error approach, but it will be very time consuming. Equation solvers such as EES are invaluable for these kinds of problems.

**8-125E**

**Solution** A water fountain is to be installed at a remote location by attaching a cast iron pipe directly to a water main. For a specified flow rate, the minimum diameter of the piping system is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed and the friction factor is constant for the entire pipe. **3** The pressure at the water main remains constant. **4** There are no dynamic pressure effects at the pipe-water main connection, and the pressure at the pipe entrance is 60 psia. **5** Elevation difference between the pipe and the fountain is negligible ($z_2 = z_1$) . **6** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Properties*** The density and dynamic viscosity of water at 70°F are $\rho$ = 62.30 lbm/ft$^3$ and $\mu$ = 2.360 lbm/ft·h = 6.556×10$^{-4}$ lbm/ft·s. The roughness of cast iron pipe is $\varepsilon$ = 0.00085 ft. The minor loss coefficient is $K_L$ = 0.5 for a sharp-edged entrance, $K_L$ = 1.1 for a 90° miter bend without vanes, $K_L$ = 0.2 for a fully open gate valve, and $K_L$ = 5 for an angle valve.

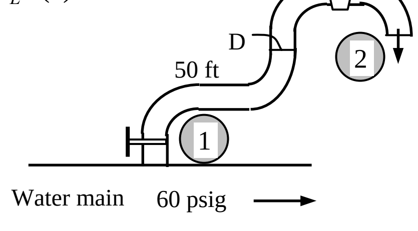
***Analysis*** We choose point 1 in the water main near the entrance where the pressure is 60 psig and the velocity in the pipe to be low. We also take point 1 as the reference level. We take point 2 at the exit of the water fountain where the pressure is the atmospheric pressure ($P_2 = P_{atm}$) and the velocity is the discharge velocity. The energy equation for a control volume between these two points is

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad \frac{P_{1,\text{gage}}}{\rho g} = \alpha_2 \frac{V_2^2}{2g} + h_L$$

where $\alpha_2$ = 1 and $h_L = h_{L,\text{total}} = h_{L,\text{major}} + h_{L,\text{minor}} = \left( f\frac{L}{D} + \sum K_L \right)\frac{V_2^2}{2g}$

since the diameter of the piping system is constant. Then the energy equation becomes

$$\frac{60 \text{ psi}}{(62.3 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)}\left(\frac{144 \text{ lbf/ft}^2}{1 \text{ psi}}\right)\left(\frac{32.2 \text{ lbm}\cdot\text{ft/s}^2}{1 \text{ lbf}}\right) = \frac{V_2^2}{2(32.2 \text{ ft/s}^2)} + h_L \quad (1)$$




The average velocity in the pipe and the Reynolds number are

$$V_2 = \frac{\dot V}{A_c} = \frac{\dot V}{\pi D^2/4} \quad \rightarrow \quad V_2 = \frac{20/60 \text{ gal/s}}{\pi D^2/4}\left(\frac{0.1337 \text{ ft}^3}{1 \text{ gal}}\right) \quad (2)$$

$$\text{Re} = \frac{\rho V_2 D}{\mu} \quad \rightarrow \quad \text{Re} = \frac{(62.3 \text{ lbm/ft}^3)V_2 D}{6.556\times10^{-4} \text{ lbm/ft}\cdot\text{s}} \quad (3)$$

The friction factor can be determined from the Colebrook equation,

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D_h}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \quad \rightarrow \quad \frac{1}{\sqrt{f}} = -2.0\log\left(\frac{0.00085/D}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}}\right) \quad (4)$$

The sum of the loss coefficients is

$$\sum K_L = K_{L,\text{entrance}} + 3K_{L,\text{elbow}} + K_{L,\text{gate valve}} + K_{L,\text{angle valve}} = 0.5 + 3\times1.1 + 0.2 + 5 = 9$$

Then the total head loss becomes

$$h_L = \left(f\frac{L}{D} + \sum K_L\right)\frac{V^2}{2g} \quad \rightarrow \quad h_L = \left(f\frac{50 \text{ ft}}{D} + 9\right)\frac{V_2^2}{2(32.2 \text{ ft/s}^2)} \quad (5)$$

These are 5 equations in the 5 unknowns of $V_2$, $h_L$, $D$, Re, and $f$, and solving them simultaneously using an equation solver such as EES gives

$V_2$ = 14.3 ft/s, $h_L$ = 135.5 ft, $D$ = 0.0630 ft = **0.76 in**, Re = 85,540, and $f$ = 0.04263

Therefore, the diameter of the pipe must be at least 0.76 in (or roughly 3/4 in).

***Discussion*** The pipe diameter can also be determined approximately by using the Swamee and Jain relation. It would give $D$ = 0.73 in, which is within 5% of the result obtained above.

**8-126**

**Solution** A water fountain is to be installed at a remote location by attaching a cast iron pipe directly to a water main. For a specified flow rate, the minimum diameter of the piping system is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed and the friction factor is constant for the entire pipe. **3** The pressure at the water main remains constant. **4** There are no dynamic pressure effects at the pipe-water main connection, and the pressure at the pipe entrance is 60 psia. **5** Elevation difference between the pipe and the fountain is negligible ($z_2 = z_1$) . **6** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Properties*** The density and dynamic viscosity of water at 70°F are $\rho$ = 62.30 lbm/ft$^3$ and $\mu$ = 2.360 lbm/ft·h = 6.556×10$^{-4}$ lbm/ft·s. The plastic pipes are considered to be smooth, and thus their roughness is $\varepsilon = 0$. The minor loss coefficient is $K_L = 0.5$ for a sharp-edged entrance, $K_L = 1.1$ for a 90° miter bend without vanes, $K_L = 0.2$ for a fully open gate valve, and $K_L = 5$ for an angle valve.

***Analysis*** We choose point 1 in the water main near the entrance where the pressure is 60 psig and the velocity in the pipe to be low. We also take point 1 as the reference level. We take point 2 at the exit of the water fountain where the pressure is the atmospheric pressure ($P_2 = P_{atm}$) and the velocity is the discharge velocity. The energy equation for a control volume between these two points is

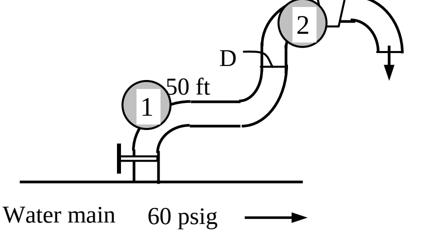
$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad \frac{P_{1,\text{gage}}}{\rho g} = \alpha_2 \frac{V_2^2}{2g} + h_L$$

where $\alpha_2 = 1$ and $h_L = h_{L,\text{total}} = h_{L,\text{major}} + h_{L,\text{minor}} = \left( f\frac{L}{D} + \sum K_L \right) \frac{V_2^2}{2g}$

since the diameter of the piping system is constant. Then the energy equation becomes

$$\frac{60 \text{ psi}}{(62.3 \text{ lbm/ft}^3)(32.2 \text{ ft/s}^2)} \left( \frac{144 \text{ lbf/ft}^2}{1 \text{ psi}} \right) \left( \frac{32.2 \text{ lbm}\cdot\text{ft/s}^2}{1 \text{ lbf}} \right) = \frac{V_2^2}{2(32.2 \text{ ft/s}^2)} + h_L \quad (1)$$

The average velocity in the pipe and the Reynolds number are




$$V_2 = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} \quad \rightarrow \quad V_2 = \frac{20/60 \text{ gal/s}}{\pi D^2/4} \left( \frac{0.1337 \text{ ft}^3}{1 \text{ gal}} \right) \quad (2)$$

$$\text{Re} = \frac{\rho V_2 D}{\mu} \quad \rightarrow \quad \text{Re} = \frac{(62.3 \text{ lbm/ft}^3) V_2 D}{6.556 \times 10^{-4} \text{ lbm/ft}\cdot\text{s}} \quad (3)$$

The friction factor can be determined from the Colebrook equation,

$$\frac{1}{\sqrt{f}} = -2.0 \log\left( \frac{\varepsilon / D_h}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}} \right) \quad \rightarrow \quad \frac{1}{\sqrt{f}} = -2.0 \log\left( 0 + \frac{2.51}{\text{Re}\sqrt{f}} \right) \quad (4)$$

The sum of the loss coefficients is

$$\sum K_L = K_{L,\text{entrance}} + 3K_{L,\text{elbow}} + K_{L,\text{gate valve}} + K_{L,\text{angle valve}} = 0.5 + 3\times 1.1 + 0.2 + 5 = 9$$

Then the total head loss becomes

$$h_L = \left( f\frac{L}{D} + \sum K_L \right) \frac{V^2}{2g} \quad \rightarrow \quad h_L = \left( f\frac{50 \text{ ft}}{D} + 9 \right) \frac{V_2^2}{2(32.2 \text{ ft/s}^2)} \quad (5)$$

These are 5 equations in the 5 unknowns of $V_2$, $h_L$, $D$, Re, and $f$, and solving them simultaneously using an equation solver such as EES gives

$V_2$ = 18.4 ft/s, $h_L$ = 133.4 ft, $D$ = 0.05549 ft = **0.67 in**, Re = 97,170, and $f$ = 0.0181

Therefore, the diameter of the pipe must be at least 0.67 in.

***Discussion*** The pipe diameter can also be determined approximately by using the Swamee and Jain relation. It would give $D$ = 0.62 in, which is within 7% of the result obtained above.

**8-127**

**Solution** In a hydroelectric power plant, the flow rate of water, the available elevation head, and the combined turbine-generator efficiency are given. The electric power output of the plant is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed and the friction factor is constant for the entire pipe. **3** The minor losses are given to be negligible. **4** The water level in the reservoir remains constant.

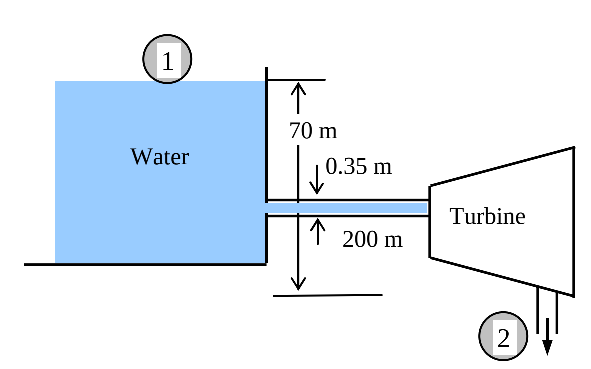
***Properties*** The density and dynamic viscosity of water at 20°C are $\rho$ = 998 kg/m$^3$ and $\mu$ = 1.002×10$^{-3}$ kg/m·s. The roughness of cast iron pipes is $\varepsilon$ = 0.00026 m.

***Analysis*** We take point 1 at the free surface of the reservoir, and point 2 and the reference level at the free surface of the water leaving the turbine site ($z_2$ = 0). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocities at both points are very low ($V_1 \cong V_2 \cong 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump,u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad h_{\text{turbine, e}}=z_1-h_L$$

The average velocity, Reynolds number, friction factor, and head loss in the pipe are

$$V=\frac{\dot V}{A_c}=\frac{\dot V}{\pi D^2/4}=\frac{0.8\text{ m}^3/\text{s}}{\pi(0.35\text{ m})^2/4}=8.315\text{ m/s}$$

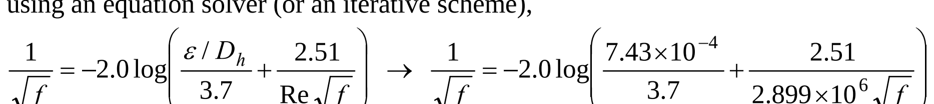



$$\text{Re}=\frac{\rho V D_h}{\mu}=\frac{(998\text{ kg/m}^3)(8.513\text{ m/s})(0.35\text{ m})}{1.002\times10^{-3}\text{ kg/m}\cdot\text{s}}=2.899\times10^6$$

which is greater than 4000. Therefore, the flow is turbulent. The relative roughness of the pipe is

$$\varepsilon/D_h=\frac{0.00026\text{ m}}{0.35\text{ m}}=7.43\times10^{-4}$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),



$$\frac{1}{\sqrt f}=-2.0\log\left(\frac{\varepsilon/D_h}{3.7}+\frac{2.51}{\text{Re}\sqrt f}\right)\quad\rightarrow\quad\frac{1}{\sqrt f}=-2.0\log\left(\frac{7.43\times10^{-4}}{3.7}+\frac{2.51}{2.899\times10^6\sqrt f}\right)$$

It gives $f$ = 0.0184. When the minor losses are negligible, the head loss in the pipe and the available turbine head are determined to be

$$h_L=f\frac{L}{D}\frac{V^2}{2g}=0.0184\frac{200\text{ m}}{0.35\text{ m}}\frac{(8.315\text{ m/s})^2}{2(9.81\text{ m/s}^2)}=37.05\text{ m}$$

$$h_{\text{turbine, e}}=z_1-h_L=70-37.05=32.95\text{ m}$$

Then the extracted power from water and the actual power output of the turbine become

$$\dot W_{\text{turbine, e}}=\dot m g h_{\text{turbine, e}}=\rho\dot V g h_{\text{turbine, e}}$$

$$=(998\text{ kg/m}^3)(0.8\text{ m}^3/\text{s})(9.81\text{ m/s}^2)(32.95\text{ m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right)\left(\frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}}\right)=258\text{ kW}$$

$$\dot W_{\text{turbine-gen}}=\eta_{\text{turbine-gen}}\dot W_{\text{turbine, e}}=(0.84)(258\text{ kW})=\mathbf{217\text{ kW}}$$

***Discussion*** Note that a perfect turbine-generator would generate 258 kW of electricity from this resource. The power generated by the actual unit is only 217 kW because of the inefficiencies of the turbine and the generator. Also note that more than half of the elevation head is lost in piping due to pipe friction.

**8-128**

**Solution** In a hydroelectric power plant, the flow rate of water, the available elevation head, and the combined turbine-generator efficiency are given. The percent increase in the electric power output of the plant is to be determined when the pipe diameter is tripled.

***Assumptions*** **1** The flow is steady and incompressible. **2** Entrance effects are negligible, and thus the flow is fully developed and friction factor is constant. **3** Minor losses are negligible. **4** Water level is constant.

***Properties*** The density and dynamic viscosity of water at 20°C are $\rho$ = 998 kg/m$^3$ and $\mu$ = 1.002×10$^{-3}$ kg/m·s. The roughness of cast iron pipes is $\varepsilon$ = 0.00026 m.

***Analysis*** We take point 1 at the free surface of the reservoir, and point 2 and the reference level at the free surface of the water leaving the turbine site ($z_2$ = 0). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocities at both points are very low ($V_1 \cong V_2 \cong 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad h_{\text{turbine, e}}=z_1-h_L$$

The average velocity, Reynolds number, friction factor, and head loss in the pipe for both cases (pipe diameter being 0.35 m and 1.05 m) are

$$V_1=\frac{\dot V}{A_{c1}}=\frac{\dot V}{\pi D_1^2/4}=\frac{0.8\text{ m}^3/\text{s}}{\pi(0.35\text{ m})^2/4}=8.315\text{ m/s},$$

$$V_2=\frac{\dot V}{A_{c2}}=\frac{\dot V}{\pi D_2^2/4}=\frac{0.8\text{ m}^3/\text{s}}{\pi(1.05\text{ m})^2/4}=0.9239\text{ m/s}$$

$$\text{Re}_1=\frac{\rho V_1 D_1}{\mu}=\frac{(998\text{ kg/m}^3)(8.315\text{ m/s})(0.35\text{ m})}{1.002\times10^{-3}\text{ kg/m}\cdot\text{s}}=2.899\times10^6$$

$$\text{Re}_2=\frac{\rho V_2 D_1}{\mu}=\frac{(998\text{ kg/m}^3)(0.9239\text{ m/s})(1.05\text{ m})}{1.002\times10^{-3}\text{ kg/m}\cdot\text{s}}=0.9662\times10^6$$



which are greater than 4000. Therefore, the flow is turbulent for both cases. The relative roughness of the pipe is

$$\varepsilon/D_1=\frac{0.00026\text{ m}}{0.35\text{ m}}=7.43\times10^{-4}\quad\text{and}\quad \varepsilon/D_2=\frac{0.00026\text{ m}}{1.05\text{ m}}=2.476\times10^{-4}$$

The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt{f_1}}=-2.0\log\left(\frac{7.43\times10^{-4}}{3.7}+\frac{2.51}{2.899\times10^6\sqrt{f_1}}\right)\quad\text{and}\quad\frac{1}{\sqrt{f_2}}=-2.0\log\left(\frac{2.476\times10^{-4}}{3.7}+\frac{2.51}{0.9662\times10^6\sqrt{f_2}}\right)$$

The friction factors are determined to be $f_1$ = 0.01842 and $f_2$ = 0.01520. When the minor losses are negligible, the head losses in the pipes and the head extracted by the turbine are determined to be

$$h_{L1}=f_1\frac{L}{D_1}\frac{V_1^2}{2g}=0.01842\frac{200\text{ m}}{0.35\text{ m}}\frac{(8.315\text{ m/s})^2}{2(9.81\text{ m/s}^2)}=37.09\text{ m},\quad h_{\text{turbine},1}=z_1-h_{L1}=70-37.09=32.91\text{ m}$$

$$h_{L2}=f_2\frac{L}{D_2}\frac{V_2^2}{2g}=0.0152\frac{200\text{ m}}{1.05\text{ m}}\frac{(0.9239\text{ m/s})^2}{2(9.81\text{ m/s}^2)}=0.13\text{ m},\quad h_{\text{turbine},2}=z_2-h_{L2}=70-0.13=69.87\text{ m}$$

The available or actual power output is proportional to the turbine head. Therefore, the increase in the power output when the diameter is doubled becomes

$$\text{Increase in power output}=\frac{h_{\text{turbine},2}-h_{\text{turbine},1}}{h_{\text{turbine},1}}=\frac{69.87-32.91}{32.91}=\mathbf{1.12\ \text{ or }\ 112\%}$$

***Discussion*** Note that the power generation of the turbine more than doubles when the pipe diameter is tripled at the same flow rate and elevation.

**8-129E**

**Solution** The drinking water needs of an office are met by siphoning water through a plastic hose inserted into a large water bottle. The time it takes to fill a glass when the bottle is first opened and when it is empty are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed and the friction factor is constant for the entire pipe. **3** The on/off switch is fully open during filing. **4** The water level in the bottle remains nearly constant during filling. **5** The flow is turbulent (to be verified). **6** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Properties*** The density and dynamic viscosity of water at 70°F are $\rho$ = 62.30 lbm/ft$^3$ and $\mu$ = 2.360 lbm/ft·h = 6.556×10$^{-4}$ lbm/ft·s. The plastic pipes are considered to be smooth, and thus their roughness is $\varepsilon = 0$. The total minor loss coefficient is given to be 2.8.

***Analysis*** We take point 1 to be at the free surface of water in the bottle, and point 2 at the exit of the hose., which is also taken to be the reference level ($z_2 = 0$). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad z_1 = \alpha_2 \frac{V_2^2}{2g} + h_L$$

where $\alpha_2 = 1$ and $h_L = \left( f\frac{L}{D} + \sum K_L \right) \frac{V_2^2}{2g} \rightarrow h_L = \left( f\frac{6 \text{ ft}}{0.35/12 \text{ ft}} + 2.8 \right) \frac{V_2^2}{2(32.2 \text{ ft/s}^2)}$ (1)

since the diameter of the piping system is constant. Then the energy equation becomes

$$z_1 = (1)\frac{V_2^2}{2(32.2 \text{ ft/s}^2)} + h_L \qquad (2)$$

The average velocity in the pipe and the Reynolds number are

$$V_2 = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi D^2/4} \rightarrow V_2 = \frac{\dot{V} \text{ ft}^3\text{/s}}{\pi (0.35/12 \text{ ft})^2/4} \qquad (3)$$

$$\text{Re} = \frac{\rho V_2 D}{\mu} \rightarrow \text{Re} = \frac{(62.3 \text{ lbm/ft}^3) V_2 (0.35/12 \text{ ft})}{6.556 \times 10^{-4} \text{ lbm/ft} \cdot \text{s}} \qquad (4)$$

The friction factor can be determined from the Colebrook equation,

$$\frac{1}{\sqrt{f}} = -2.0 \log \left( \frac{\varepsilon / D_h}{3.7} + \frac{2.51}{\text{Re}\sqrt{f}} \right) \rightarrow \frac{1}{\sqrt{f}} = -2.0 \log \left( 0 + \frac{2.51}{\text{Re}\sqrt{f}} \right) \qquad (5)$$

Finally, the filling time of the glass is

$$\Delta t = \frac{V_{\text{glass}}}{\dot{V}} = \frac{0.00835 \text{ ft}^3}{\dot{V} \text{ ft}^3\text{/s}} \qquad (6)$$

6 ft
0.35
1 ft
3 ft
1
2

These are 6 equations in the 6 unknowns of $V_2$, $\dot{V}$, $h_L$, Re, $f$, and $\Delta t$, and solving them simultaneously using an equation solver such as EES with the appropriate $z_1$ value gives

**Case (*a*):** The bottle is full and thus $z_1$ = 3+1 = 4 ft:

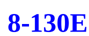
$V_2$ = 5.185 ft/s, $h_L$ = 3.58 ft, $\dot{V}$ = 0.00346 ft$^3$/s, Re = 14,370, $f$ = 0.02811, and $\Delta t$ = **2.4 s**

**Case (*b*):** The bottle is almost empty and thus $z_1$ = 3 ft:

$V_2$ =4.436 ft/s, $h_L$ = 2.69 ft, $\dot{V}$ = 0.00296 ft$^3$/s, Re = 12,290, $f$ = 0.02926, and $\Delta t$ = **2.8 s**

Note that the flow is turbulent for both cases since Re > 4000.

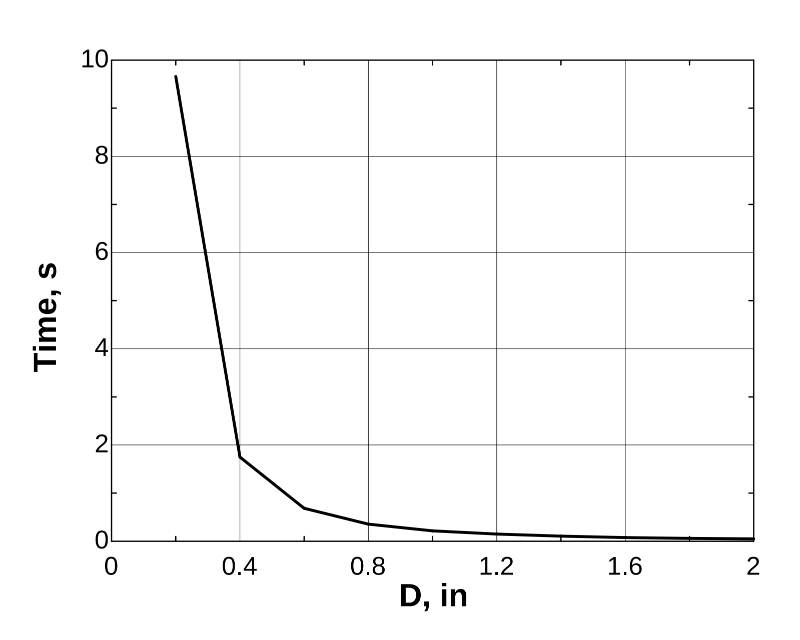
***Discussion*** The filling time of the glass increases as the water level in the bottle drops, as expected.


**8-130E** 

**Solution** In the previous problem, the effect of the hose diameter on the time required to fill a glass when the bottle is full is to be investigated by varying the pipe diameter from 0.2 to 2 in. in increments of 0.2 in.

***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
rho=62.3
mu=2.36/3600
nu=mu/rho
g=32.2
z1=4
Volume=0.00835
D=Din/12
Ac=pi*D^2/4
L=6
KL=2.8
eps=0
rf=eps/D
V=Vdot/Ac
"Reynolds number"
Re=V*D/nu
1/sqrt(f)=-2*log10(rf/3.7+2.51/(Re*sqrt(f)))
HL=(f*L/D+KL)*(V^2/(2*g))
z1=V^2/(2*g)+HL
Time=Volume/Vdot
```




| *D*, in | *Time,* s | $h_L$, ft | Re |
|---|---|---|---|
| 0.2 | 9.66 | 3.76 | 6273 |
| 0.4 | 1.75 | 3.54 | 17309 |
| 0.6 | 0.68 | 3.40 | 29627 |
| 0.8 | 0.36 | 3.30 | 42401 |
| 1.0 | 0.22 | 3.24 | 55366 |
| 1.2 | 0.15 | 3.20 | 68418 |
| 1.4 | 0.11 | 3.16 | 81513 |
| 1.6 | 0.08 | 3.13 | 94628 |
| 1.8 | 0.06 | 3.11 | 107752 |
| 2.0 | 0.05 | 3.10 | 120880 |

***Discussion*** The required time decreases considerably as the tube diameter increases. This is because the irreversible frictional head loss (major loss) in the tube decreases greatly as tube diameter increases. In addition, the minor loss is proportional to $V^2$. Thus, as tube diameter increases, $V$ decreases, and even the minor losses decrease.

**8-131E**

**Solution** The drinking water needs of an office are met by siphoning water through a plastic hose inserted into a large water bottle. The time it takes to fill a glass when the bottle is first opened is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The entrance effects are negligible, and thus the flow is fully developed and the friction factor is constant for the entire pipe. **3** The on/off switch is fully open during filing. **4** The water level in the bottle remains constant during filling. **5** The flow is turbulent (to be verified). **6** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Properties*** The density and dynamic viscosity of water at 70°F are $\rho$ = 62.30 lbm/ft$^3$ and $\mu$ = 2.360 lbm/ft·h = 6.556×10$^{-4}$ lbm/ft·s. The plastic pipes are considered to be smooth, and thus their roughness is $\varepsilon = 0$. The total minor loss coefficient is given to be 2.8 during filling.

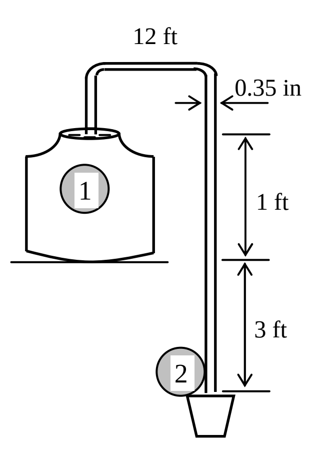
***Analysis*** We take point 1 to be at the free surface of water in the bottle, and point 2 at the exit of the hose, which is also taken to be the reference level ($z_2 = 0$). Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad z_1=\alpha_2\frac{V_2^2}{2g}+h_L$$

where $\alpha_2 = 1$ and
$$h_L=\left(f\frac{L}{D}+\sum K_L\right)\frac{V_2^2}{2g}\quad\rightarrow\quad h_L=\left(f\frac{12\text{ ft}}{0.35/12\text{ ft}}+2.8\right)\frac{V_2^2}{2(32.2\text{ ft/s}^2)} \qquad (1)$$

since the diameter of the piping system is constant. Then the energy equation becomes

$$z_1=(1)\frac{V_2^2}{2(32.2\text{ ft/s}^2)}+h_L \qquad (2)$$




The average velocity in the pipe and the Reynolds number are

$$V_2=\frac{\dot V}{A_c}=\frac{\dot V}{\pi D^2/4}\quad\rightarrow\quad V_2=\frac{\dot V\text{ ft}^3\text{/s}}{\pi(0.35/12\text{ ft})^2/4} \qquad (3)$$

$$\text{Re}=\frac{\rho V_2 D}{\mu}\quad\rightarrow\quad \text{Re}=\frac{(62.3\text{ lbm/ft}^3)V_2(0.35/12\text{ ft})}{1.307\times10^{-3}\text{ lbm/ft}\cdot\text{s}} \qquad (4)$$

The friction factor can be determined from the Colebrook equation,

$$\frac{1}{\sqrt f}=-2.0\log\left(\frac{\varepsilon/D_h}{3.7}+\frac{2.51}{\text{Re}\sqrt f}\right)\quad\rightarrow\quad \frac{1}{\sqrt f}=-2.0\log\left(0+\frac{2.51}{\text{Re}\sqrt f}\right) \qquad (5)$$

Finally, the filling time of the glass is

$$\Delta t=\frac{V_{\text{glass}}}{\dot V}=\frac{0.00835\text{ ft}^3}{\dot V\text{ ft}^3\text{/s}} \qquad (6)$$

These are 6 equations in the 6 unknowns of $V_2$, $\dot V$, $h_L$, Re, $f$, and $\Delta t$, and solving them simultaneously using an equation solver such as EES with the appropriate $z_1$ value gives

**Case (*a*):** The bottle is full and thus $z_1$ = 3+1 = 4 ft:

$V_2$ =3.99 ft/s, $h_L$ = 3.75 ft, $\dot V$ = 0.002667 ft$^3$/s, Re = 11,060, $f$ = 0.03007, and $\Delta t$ = **3.1 s**

**Case (*b*):** The bottle is almost empty and thus $z_1$ = 3 ft:

$V_2$ =3.40 ft/s, $h_L$ = 2.82 ft, $\dot V$ = 0.002272 ft$^3$/s, Re = 9426, $f$ =0.03137, and $\Delta t$ = **3.7 s**

Note that the flow is turbulent for both cases since Re > 4000.

***Discussion*** The filling times in Prob. 8-129E were 2.4 s and 2.8 s, respectively. Therefore, doubling the tube length increases the filling time by 0.7 s when the bottle is full, and by 0.9 s when it is empty.

**8-132**

**Solution** A water pipe has an abrupt expansion from diameter $D_1$ to $D_2$. It is to be shown that the loss coefficient is $K_L = (1 - D_1^2 / D_2^2)^2$, and $K_L$ and $P_2$ are to be calculated.

***Assumptions*** **1** The flow is steady and incompressible. **2** The pressure is uniform at the cross-section where expansion occurs, and is equal to the upstream pressure $P_1$. **3** The flow section is horizontal (or the elevation difference across the expansion section is negligible). **4** The flow is turbulent, and the effects of kinetic energy and momentum-flux correction factors are negligible, $\beta \approx 1$ and $\alpha \approx 1$.

***Properties*** We take the density of water to be $\rho = 1000$ kg/m$^3$.

***Analysis*** We designate the cross-section where expansion occurs by $x$. We choose cross-section 1 in the smaller diameter pipe shortly before $x$, and section 2 in the larger diameter pipe shortly after $x$. We take the region occupied by the fluid between cross-sections 1 and 2 as the control volume, with an inlet at 1 and exit at 2. The velocity, pressure, and cross-sectional area are $V_1$, $P_1$, and $A_1$ at cross-section 1, and $V_2$, $P_2$, and $A_2$ at cross-section 2. We assume the pressure along the cross-section $x$ to be $P_1$ so that $P_x = P_1$. Then the continuity, momentum, and energy equations applied to the control volume become

(1) *Continuity*: $\dot m_1 = \dot m_2 \;\rightarrow\; \rho V_1 A_1 = \rho V_2 A_2 \;\rightarrow\; V_2 = \dfrac{A_1}{A_2} V_1$ (1)

(2) *Momentum*: $\sum \vec F = \sum_{\text{out}} \beta \dot m V - \sum_{\text{in}} \beta \dot m V \;\rightarrow\; P_1 A_1 + P_1(A_x - A_1) - P_2 A_2 = \dot m (V_2 - V_1)$

But $P_1 A_1 + P_1(A_x - A_1) = P_1 A_x = P_1 A_2$

$$\dot m (V_2 - V_1) = \rho A_2 V_2 (V_2 - V_1) = \rho A_2 \frac{A_1}{A_2} V_1 \left( \frac{A_1}{A_2} V_1 - V_1 \right) = \rho A_2 \frac{A_1}{A_2}\left( \frac{A_1}{A_2} - 1 \right) V_1^2$$

Therefore, $P_1 A_2 - P_2 A_2 = \rho A_2 \dfrac{A_1}{A_2}\left(\dfrac{A_1}{A_2} - 1\right) V_1^2 \;\rightarrow\; \dfrac{P_1 - P_2}{\rho} = \dfrac{A_1}{A_2}\left(\dfrac{A_1}{A_2} - 1\right) V_1^2$ (2)

(3) *Energy*: $\dfrac{P_1}{\rho g} + \alpha_1 \dfrac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \dfrac{P_2}{\rho g} + \alpha_2 \dfrac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \;\rightarrow\; h_L = \dfrac{P_1 - P_2}{\rho g} + \dfrac{V_1^2 - V_2^2}{2g}$ (3)

Substituting Eqs. (1) and (2) and $h_L = K_L \dfrac{V_1^2}{2g}$ into Eq. (3) gives

$$K_L \frac{V_1^2}{2g} = \frac{A_1}{A_2}\left(\frac{A_1}{A_2} - 1\right)\frac{V_1^2}{g} + \frac{V_1^2 - (A_1^2 / A_2^2) V_1^2}{2g} \;\rightarrow\; K_L = \frac{2A_1}{A_2}\left(\frac{A_1}{A_2} - 1\right) + \left(1 - \frac{A_1^2}{A_2^2}\right)$$

Simplifying and substituting $A = \pi D^2/4$ gives the desired relation and its value,

$$K_L = \left(1 - \frac{A_1}{A_1}\right)^2 = \left(1 - \frac{\pi D_1^2/4}{\pi D_2^2/4}\right)^2 = \left(1 - \frac{D_1^2}{D_2^2}\right)^2 = \left(1 - \frac{(0.15\text{ m})^2}{(0.20\text{ m})^2}\right)^2 = 0.1914$$

Also, $h_L = K_L \dfrac{V_1^2}{2g} = (0.1914)\dfrac{(10\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 0.9756\text{ m}$

$$V_2 = \frac{A_1}{A_2} V_1 = \frac{D_1^2}{D_2^2} V_1 = \frac{(0.15\text{ m})^2}{(0.20\text{ m})^2}(10\text{ m/s}) = 5.625\text{ m/s}$$



Solving for $P_2$ from Eq. (3) and substituting,

$$P_2 = P_1 + \rho\{(V_1^2 - V_2^2)/2 - g h_L\}$$

$$= (120\text{ kPa}) + (1000\text{ kg/m}^3)\left\{\frac{(10\text{ m/s})^2 - (5.625\text{ m/s})^2}{2} - (9.81\text{ m/s}^2)(0.9756\text{ m})\right\}\left(\frac{1\text{ kPa}\cdot\text{m}^2}{1000\text{ kg}\cdot\text{m/s}^2}\right) = \mathbf{145\text{ kPa}}$$

Note that the pressure increases by 25 kPa after the expansion due to the conversion of dynamic pressure to static pressure when the velocity is decreased. Also, $K_L \cong 1$ (actually, $K_L = \alpha$) when $D_2 >> D_1$ (discharging into a reservoir).

***Discussion*** At a discharge into a large reservoir, *all* the kinetic energy is wasted as heat.

**8-133**

**Solution** A swimming pool is initially filled with water. A pipe with a well-rounded entrance at the bottom drains the pool to the atmosphere. The initial rate of discharge from the pool and the time required to empty the pool completely are to be determined.

***Assumptions*** **1** The flow is uniform and incompressible. **2** The draining pipe is horizontal. **3** The entrance effects are negligible, and thus the flow is fully developed. **4** The friction factor remains constant (in reality, it changes since the flow velocity and thus the Reynolds number changes during flow). **5** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

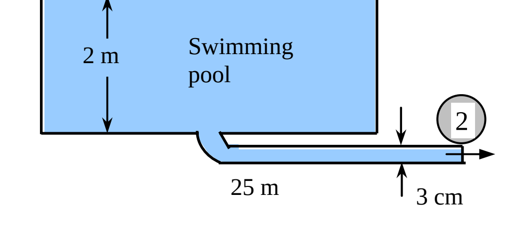
***Properties*** The density and dynamic viscosity of water at 20°C are $\rho$ = 998 kg/m$^3$ and $\mu$ = 1.002×10$^{-3}$ kg/m·s. The friction factor of the pipe is given to be 0.022. Plastic pipes are considered to be smooth, and their surface roughness is $\varepsilon$ = 0.

***Analysis*** We take point 1 at the free surface of the pool, and point 2 and the reference level at the exit of the pipe ($z_2$ = 0), and take the positive direction of $z$ to be upwards. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1+h_{\text{pump, u}}=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_{\text{turbine, e}}+h_L \quad\rightarrow\quad z_1=\alpha_2\frac{V_2^2}{2g}+h_L$$

where

$$h_L = f\frac{L}{D}\frac{V^2}{2g}$$




since the minor losses are negligible. Substituting and solving for $V_2$ gives

$$z_1=\alpha_2\frac{V_2^2}{2g}+\left(f\frac{L}{D}\right)\frac{V_2^2}{2g} \quad\rightarrow\quad V_2=\sqrt{\frac{2gz_1}{\alpha_2+fL/D}}$$

Noting that $\alpha_2$ = 1 and initially $z_1$ = 2 m, the initial velocity and flow rate are determined to be

$$V_{2,i}=\sqrt{\frac{2gz_1}{1+fL/D}}=\sqrt{\frac{2(9.81\text{ m/s}^2)(2\text{ m})}{1+0.022(25\text{ m})/(0.03\text{ m})}}=1.425\text{ m/s}$$

$$\dot{V}_{initial}=V_{2,i}A_c=V_{2,i}(\pi D^2/4)=(1.425\text{ m/s})[\pi(0.03\text{ m})^2/4]=1.01\times10^{-3}\text{ m}^3\text{/s}=1.01\text{ L/s}$$

The average discharge velocity at any given time, in general, can be expressed as

$$V_2=\sqrt{\frac{2gz}{1+fL/D}}$$

where $z$ is the water height relative to the center of the orifice at that time.

We denote the diameter of the pipe by $D$, and the diameter of the pool by $D_o$. The flow rate of water from the pool can be obtained by multiplying the discharge velocity by the pipe cross-sectional area,

$$\dot{V}=A_cV_2=\frac{\pi D^2}{4}\sqrt{\frac{2gz}{1+fL/D}}$$

Then the amount of water that flows through the pipe during a differential time interval $dt$ is

$$dV=\dot{V}dt=\frac{\pi D^2}{4}\sqrt{\frac{2gz}{1+fL/D}}dt \qquad (1)$$

which, from conservation of mass, must be equal to the decrease in the volume of water in the pool,

$$dV=A_{c,\tan k}(-dz)=-\frac{\pi D_0^2}{4}dz \qquad (2)$$

where $dz$ is the change in the water level in the pool during $dt$. (Note that $dz$ is a negative quantity since the positive direction of $z$ is upwards. Therefore, we used $-dz$ to get a positive quantity for the amount of water discharged). Setting Eqs. (1) and (2) equal to each other and rearranging,

$$\frac{\pi D^2}{4}\sqrt{\frac{2gz}{1+fL/D}}dt = -\frac{\pi D_0^2}{4}dz \;\rightarrow\; dt = -\frac{D_0^2}{D^2}\sqrt{\frac{1+fL/D}{2gz}}dz = -\frac{D_0^2}{D^2}\sqrt{\frac{1+fL/D}{2g}}z^{-\frac{1}{2}}dz$$

The last relation can be integrated easily since the variables are separated. Letting $t_f$ be the discharge time and integrating it from $t = 0$ when $z = z_1$ to $t = t_f$ when $z = 0$ (completely drained pool) gives

$$\int_{t=0}^{t_f} dt = -\frac{D_0^2}{D^2}\sqrt{\frac{1+fL/D}{2g}}\int_{z=z_1}^{0} z^{-1/2}dz \;\rightarrow\; t_f = -\frac{D_0^2}{D^2}\sqrt{\frac{1+fL/D}{2g}}\left.\frac{z^{\frac{1}{2}}}{\frac{1}{2}}\right|_{z_1}^{0} = \frac{2D_0^2}{D^2}\sqrt{\frac{1+fL/D}{2g}}z_1^{\frac{1}{2}}$$

Simplifying and substituting the values given, the draining time is determined to be

$$t_f = \frac{D_0^2}{D^2}\sqrt{\frac{2z_1(1+fL/D)}{g}} = \frac{(10\text{ m})^2}{(0.03\text{ m})^2}\sqrt{\frac{2(2\text{ m})[1+(0.022)(25\text{ m})/(0.03\text{ m})]}{9.81\text{ m/s}^2}} = 312{,}000\text{ s} = \mathbf{86.7\ h}$$

**Checking:** For plastic pipes, the surface roughness and thus the roughness factor is zero. The Reynolds number at the beginning of draining process is

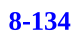
$$\text{Re} = \frac{\rho V_2 D}{\mu} = \frac{(998\text{ kg/m}^3)(1.425\text{ m/s})(0.03\text{ m})}{1.002\times 10^{-3}\text{ kg/m}\cdot\text{s}} = 42{,}580$$

which is greater than 4000. The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7}+\frac{2.51}{\text{Re}\sqrt{f}}\right) \;\rightarrow\; \frac{1}{\sqrt{f}} = -2.0\log\left(0+\frac{2.51}{42{,}570\sqrt{f}}\right)$$

It gives $f = 0.022$. Therefore, the given value of 0.022 is accurate.

***Discussion*** It can be shown by setting $L = 0$ that the draining time without the pipe is only about 18 h. Therefore, the pipe in this case increases the draining time by about a factor of 5.



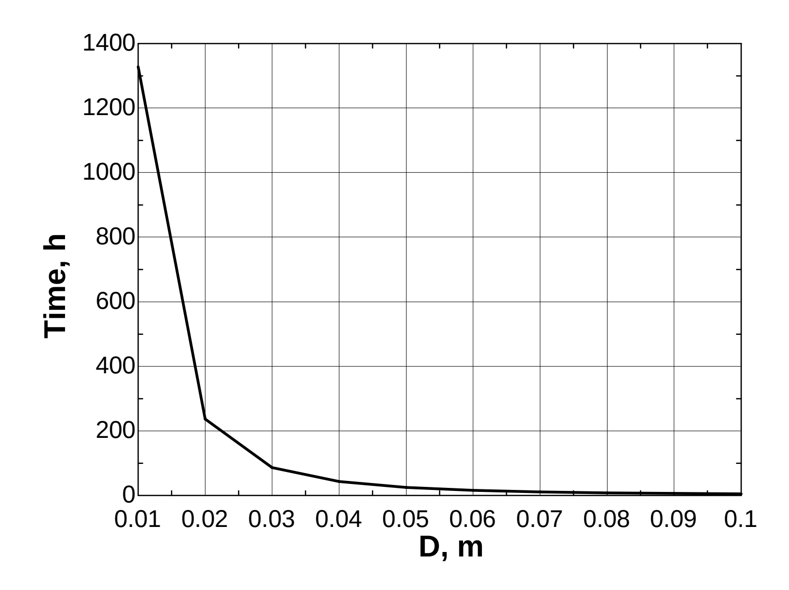
**8-134** 

**Solution** In the previous problem, the effect of the discharge pipe diameter on the time required to empty the pool completely is to be investigated by varying the pipe diameter from 1 cm to 10 cm in increments of 1 cm.

***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
rho=998
mu=0.001002
g=9.81
Dtank= 10
Ac=pi*D^2/4
L=25
f=0.022
z1=2
V=(2*g*z1/(1+f*L/D))^0.5
Vdot=V*Ac
Time=(Dtank/D)^2*(2*z1*(1+f*L/D)/g)^0.5/3600
```

| *D*, m | *Time*, h | $V_{\text{initial}}$, m/s | Re |
|---|---|---|---|
| 0.01 | 1327.4 | 0.84 | 8337 |
| 0.02 | 236.7 | 1.17 | 23374 |
| 0.03 | 86.7 | 1.42 | 42569 |
| 0.04 | 42.6 | 1.63 | 64982 |
| 0.05 | 24.6 | 1.81 | 90055 |
| 0.06 | 15.7 | 1.96 | 117406 |
| 0.07 | 10.8 | 2.10 | 146750 |
| 0.08 | 7.8 | 2.23 | 177866 |
| 0.09 | 5.8 | 2.35 | 210572 |
| 0.10 | 4.5 | 2.46 | 244721 |




***Discussion*** The required drain time decreases quite rapidly as pipe diameter is increased.

## 8-135

**Solution** A swimming pool is initially filled with water. A pipe with a sharp-edged entrance at the bottom drains the pool to the atmosphere. The initial rate of discharge from the pool and the time required to empty the pool completely are to be determined.

***Assumptions*** **1** The flow is uniform and incompressible. **2** The draining pipe is horizontal. **3** The flow is turbulent so that the tabulated value of the loss coefficient can be used. **4** The friction factor remains constant (in reality, it changes since the flow velocity and thus the Reynolds number changes during flow). **5** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Properties*** The density and dynamic viscosity of water at 20°C are $\rho = 998$ kg/m$^3$ and $\mu = 1.002\times10^{-3}$ kg/m·s. The loss coefficient for the sharp-edged entrance is $K_L = 0.5$. Plastic pipes are considered to be smooth, and their surface roughness is $\varepsilon = 0$.

***Analysis*** We take point 1 at the free surface of the pool, and point 2 and the reference level at the exit of the pipe ($z_2 = 0$), and take the positive direction of $z$ to be upwards. Noting that the fluid at both points is open to the atmosphere (and thus $P_1 = P_2 = P_{\text{atm}}$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the energy equation for a control volume between these two points (in terms of heads) simplifies to

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 + h_{\text{pump, u}} = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_{\text{turbine, e}} + h_L \quad \rightarrow \quad z_1 = \alpha_2 \frac{V_2^2}{2g} + h_L$$

where $\alpha_2 = 1$ and

$$h_L = h_{L,\text{total}} = h_{L,\text{major}} + h_{L,\text{minor}} = \left(f\frac{L}{D} + \sum K_L\right)\frac{V^2}{2g} = \left(f\frac{L}{D} + K_L\right)\frac{V_2^2}{2g}$$

since the diameter of the piping system is constant. Substituting and solving for $V_2$ gives

$$z_1 = \frac{V_2^2}{2g} + \left(f\frac{L}{D} + K_L\right)\frac{V_2^2}{2g} \quad \rightarrow \quad V_2 = \sqrt{\frac{2gz_1}{1 + fL/D + K_L}}$$

Noting that initially $z_1 = 2$ m, the initial velocity and flow rate are determined to be

$$V_{2,i} = \sqrt{\frac{2gz_1}{1 + fL/D + K_L}} = \sqrt{\frac{2(9.81\ \text{m/s}^2)(2\ \text{m})}{1 + 0.022(25\ \text{m})/(0.03\ \text{m}) + 0.5}} = 1.407\ \text{m/s}$$

$$\dot{V}_{initial} = V_{2,i}A_c = V_{2,i}(\pi D^2/4) = (1.407\ \text{m/s})[\pi(0.03\ \text{m})^2/4] = 9.94\times10^{-4}\ \text{m}^3/\text{s} = 0.994\ \text{L/s}$$

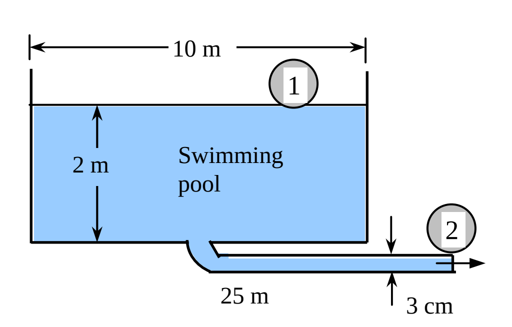
The average discharge velocity at any given time, in general, can be expressed as

$$V_2 = \sqrt{\frac{2gz}{1 + fL/D + K_L}}$$

where $z$ is the water height relative to the center of the orifice at that time.

We denote the diameter of the pipe by $D$, and the diameter of the pool by $D_o$. The flow rate of water from the pool can be obtained by multiplying the discharge velocity by the pipe cross-sectional area,

$$\dot{V} = A_c V_2 = \frac{\pi D^2}{4}\sqrt{\frac{2gz}{1 + fL/D + K_L}}$$




Then the amount of water that flows through the pipe during a differential time interval $dt$ is

$$dV = \dot{V}dt = \frac{\pi D^2}{4}\sqrt{\frac{2gz}{1 + fL/D + K_L}}\,dt \qquad (1)$$

which, from conservation of mass, must be equal to the decrease in the volume of water in the pool,

$$dV = A_{c,\tan k}(-dz) = -\frac{\pi D_0^2}{4}dz \qquad (2)$$

where $dz$ is the change in the water level in the pool during $dt$. (Note that $dz$ is a negative quantity since the positive direction of $z$ is upwards. Therefore, we used $-dz$ to get a positive quantity for the amount of water discharged). Setting Eqs. (1) and (2) equal to each other and rearranging,

$$\frac{\pi D^2}{4}\sqrt{\frac{2gz}{1+fL/D+K_L}}dt=-\frac{\pi D_0^2}{4}dz \rightarrow dt=-\frac{D_0^2}{D^2}\sqrt{\frac{1+fL/D+K_L}{2gz}}dz=-\frac{D_0^2}{D^2}\sqrt{\frac{1+fL/D+K_L}{2g}}z^{-\frac{1}{2}}dz$$

The last relation can be integrated easily since the variables are separated. Letting $t_f$ be the discharge time and integrating it from $t = 0$ when $z = z_1$ to $t = t_f$ when $z = 0$ (completely drained pool) gives

$$\int_{t=0}^{t_f}dt=-\frac{D_0^2}{D^2}\sqrt{\frac{1+fL/D+K_L}{2g}}\int_{z=z_1}^{0}z^{-1/2}dz \rightarrow t_f=-\frac{D_0^2}{D^2}\sqrt{\frac{1+fL/D+K_L}{2g}}\left.\frac{z^{\frac{1}{2}}}{\frac{1}{2}}\right|_{z_1}^{0}=\frac{2D_0^2}{D^2}\sqrt{\frac{1+fL/D+K_L}{2g}}z_1^{\frac{1}{2}}$$

Simplifying and substituting the values given, the draining time is determined to be

$$t_f=\frac{D_0^2}{D^2}\sqrt{\frac{2z_1(1+fL/D+K_L)}{g}}=\frac{(10\text{ m})^2}{(0.03\text{ m})^2}\sqrt{\frac{2(2\text{ m})[1+(0.022)(25\text{ m})/(0.03\text{ m})+0.5]}{9.81\text{ m/s}^2}}=316{,}000\text{ s}=\mathbf{87.8\ h}$$

This is a change of (87.8-86.7)/86.7 = 0.013 or 1.3%. Therefore, the minor loss in this case is **truly minor**.

**Checking:** For plastic pipes, the surface roughness and thus the roughness factor is zero. The Reynolds number at the beginning of draining process is

$$\text{Re}=\frac{\rho V_2 D}{\mu}=\frac{(998\text{ kg/m}^3)(1.407\text{ m/s})(0.03\text{ m})}{1.002\times10^{-3}\text{ kg/m}\cdot\text{s}}=42{,}030$$

which is greater than 4000. The friction factor can be determined from the Moody chart, but to avoid the reading error, we determine it from the Colebrook equation using an equation solver (or an iterative scheme),

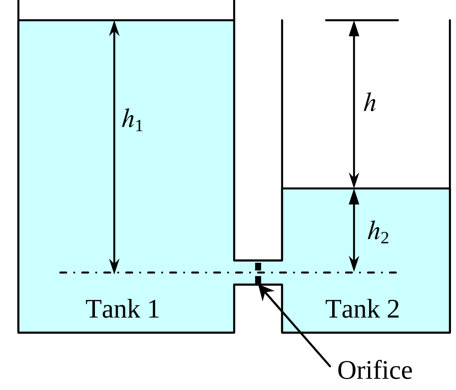
$$\frac{1}{\sqrt{f}}=-2.0\log\left(\frac{\varepsilon/D}{3.7}+\frac{2.51}{\text{Re}\sqrt{f}}\right) \rightarrow \frac{1}{\sqrt{f}}=-2.0\log\left(0+\frac{2.51}{42{,}030\sqrt{f}}\right)$$

It gives $f = 0.022$. Therefore, the given value of 0.022 is accurate.

***Discussion*** It can be shown by setting $L = 0$ that the draining time without the pipe is only about 24 h. Therefore, the pipe in this case increases the draining time more than 3 folds.

**8-136**

**Solution** A system that concsists of two interconnected cylindrical tanks is used to determine the discharge coefficient of a short 5-mm diameter orifice. For given initial fluid heights and discharge time, the discharge coefficient of the orifice is to be determined.




***Assumptions*** **1** The fluid is incompressible. **2** The entire systems, including the connecting flow section, is horizontal. **3** The discharge coefficient remains constant (in reality, it may change since the flow velocity and thus the Reynolds number changes during flow). **4** Losses other than the ones associated with flow through the orifice are negligible. **5** The effect of the kinetic energy correction factor is negligible, $\alpha = 1$.

***Analysis*** We take point 1 at the free surface of water in Tank 1, and point 0 at the exit of the orifice. We take the centerline of the orifice as the reference level ($z_1 = h_1$ and $z_0 = 0$). Noting that the fluid at point 1is open to the atmosphere (and thus $P_1 = P_{atm}$ and $P_0 = P_{atm} + \rho g h_2$) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$), the *Bernoulli equation* between these two points gives

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{P_0}{\rho g} + \frac{V_0^2}{2g} + z_0 \;\rightarrow\; \frac{P_{atm}}{\rho g} + h_1 = \frac{P_{atm} + \rho g h_2}{\rho g} + \frac{V_0^2}{2g} \;\rightarrow\; V_0 = \sqrt{2g(h_1 - h_2)} = \sqrt{2gh}$$

where $h = h_1 - h_2$ is the vertical distance between the water levels in the two tanks at any time $t$. Note that $h_1$, $h_2$, $h$, and $V_0$ are all variable ($h_1$ decreases while $h_2$ and $h$ increase during discharge.

Noting that the fluid is a liquid (ρ = constant) and keeping the conservation of mass in mind and the definition of the discharge coefficient $C_d$, the flow rate through the orifice can be expressed as

$$\dot{V} = C_d V_o A_o = -A_1 \frac{dh_1}{dt} = A_2 \frac{dh_2}{dt} \longrightarrow dh_2 = -\frac{A_1}{A_2} dh_1$$

Also, $h = h_1 - h_2 \longrightarrow dh = dh_1 - dh_2 \longrightarrow dh_1 = dh_2 + dh$ (Note that $dh < 0$, $dh_1 < 0$, and $dh_2 > 0$)

Combining the two equations above, $$dh_1 = \frac{dh}{1 + A_1/A_2}$$

Then, $$\dot{V} = C_d V_o A_o = -A_1 \frac{dh_1}{dt} \longrightarrow C_d A_o \sqrt{2gh} = -A_1 \frac{1}{1 + A_1/A_2} \frac{dh}{dt}$$

which can be rearranged as $$-dt = \frac{A_1 A_2}{A_1 + A_2} \frac{1}{C_d A_o \sqrt{2g}} \frac{dh}{\sqrt{h}}$$

Integrating $$-\int_0^t dt = \frac{A_1 A_2}{A_1 + A_2} \frac{1}{C_d A_o \sqrt{2g}} \int_{h_1}^{h} \frac{dh}{\sqrt{h}}$$

Performing the integration $$t = -\frac{A_1 A_2}{A_1 + A_2} \frac{2}{C_d A_o \sqrt{2g}} \left[\sqrt{h} - \sqrt{h_1}\right]$$

Solving for $C_d$ $$C_d = \frac{2(\sqrt{h_1} - \sqrt{h})}{(A_0/A_2 + A_0/A_1) t \sqrt{2g}}$$

Fluid flow stops when the liquid levels in the two tanks become equal (and thus $h = 0$). Substituign the given values, the discharge coefficient is determined to be

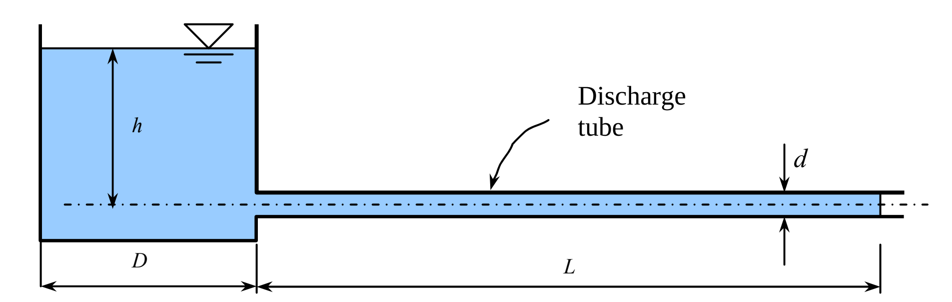
$$\frac{A_0}{A_2} + \frac{A_0}{A_1} = \left(\frac{D_0}{D_2}\right)^2 + \left(\frac{D_0}{D_1}\right)^2 = \left(\frac{0.5\text{ cm}}{30\text{ cm}}\right)^2 + \left(\frac{0.5\text{ cm}}{12\text{ cm}}\right)^2 = 0.002014\,,$$

$$C_d = \frac{2\sqrt{0.5\text{ m}}}{(0.002014)(170\text{ s})\sqrt{2 \times 9.81\text{ m/s}^2}} = \mathbf{0.933}$$

***Discussion*** We could add the minor losses at the pipe inlet and outlet without much extra effort.

**8-137**

**Solution** A highly viscous liquid discharges from a large container through a small diameter tube in laminar flow. A relation is to be obtained for the variation of fluid depth in the tank with time.




***Assumptions*** **1** The fluid is incompressible. **2** The discharge tube is horizontal, and the flow is laminar. **3** Entrance effects and the velocity heads are negligible.

***Analysis*** We take point 1 at the free surface of water in the tank, and point 2 at the exit of the pipe. We take the centerline of the pipe as the reference level ($z_1 = h$ and $z_2 = 0$). Noting that the fluid at both points 1 and 2 are open to the atmosphere (and thus $P_1 = P_2 = P_{atm}$ ) and that the fluid velocity at the free surface is very low ($V_1 \cong 0$) and the velocity heads are negligible, the energy equation for a control volume between these two points gives

$$\frac{P_1}{\rho g} + \alpha_1 \frac{V_1^2}{2g} + z_1 = \frac{P_2}{\rho g} + \alpha_2 \frac{V_2^2}{2g} + z_2 + h_L \quad \rightarrow \quad \frac{P_{atm}}{\rho g} + h = \frac{P_{atm}}{\rho g} + h_L \quad \rightarrow \quad h_L = h \qquad (1)$$

where $h$ is the liquid height in the tank at any time $t$. The total head loss through the pipe consists of major losses in the pipe since the minor losses are negligible. Also, the entrance effects are negligible and thus the friction factor for the entire tube is constant at the fully developed value. Noting that $f = \text{Re}/64$ for fully developed laminar flow in a circular pipe of diameter $d$, the head loss can be expressed as

$$h_L = f \frac{L}{d} \frac{V^2}{2g} = \frac{64}{\text{Re}} \frac{L}{d} \frac{V^2}{2g} = \frac{64}{Vd/\nu} \frac{L}{d} \frac{V^2}{2g} = \frac{64\nu L}{d^2} \frac{V}{2g} \qquad (2)$$

The average velocity can be expressed in terms of the flow rate as $V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{\pi d^2/4}$. Substituting into (2),

$$h_L = \frac{64\nu L}{d^2} \frac{1}{2g} \left( \frac{\dot{V}}{\pi d^2/4} \right) = \frac{64\nu L}{d^2} \frac{4\dot{V}}{2g\pi d^2} = \frac{128\nu L\dot{V}}{g\pi d^4} \qquad (3)$$

Combining Eqs. (1) and (3): $$h = \frac{128\nu L\dot{V}}{g\pi d^4} \qquad (4)$$

Noting that the liquid height $h$ in the tank decreases during flow, the flow rate can also be expressed in terms of the rate of change of liquid height in the tank as

$$\dot{V} = -A_{\text{tank}} \frac{dh}{dt} = -\frac{\pi D^2}{4} \frac{dh}{dt} \qquad (5)$$

Substituting Eq. (5) into (4): $$h = -\frac{128\nu L}{g\pi d^4} \frac{\pi D^2}{4} \frac{dh}{dt} = -\frac{32\nu LD^2}{gd^4} \frac{dh}{dt} \qquad (6)$$

To separate variables, it can be rearranged as $$dt = -\frac{32\nu LD^2}{gd^4} \frac{dh}{h}$$

Integrating from $t = 0$ (at which $h = H$) to $t = t$ (at which $h = h$) gives

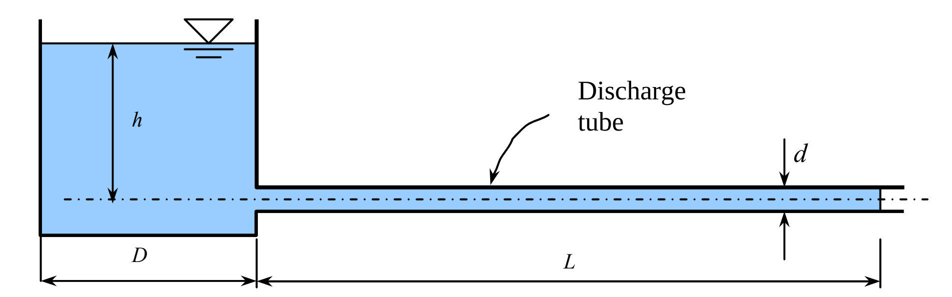
$$\boxed{t = \frac{32\nu LD^2}{gd^4} \ln(H/h)}$$

which is the desired relation for the variation of fluid depth $h$ in the tank with time $t$.

***Discussion*** If the entrance effects and the outlet kinetic energy were included in the analysis, the time would be slower.

**8-138**

**Solution** Using the setup described in the previous problem, the viscosity of an oil is to be determined for a given set of data.




***Assumptions*** **1** The oil is incompressible. **2** The discharge tube is horizontal, and the flow is laminar. **3** Entrance effects and the inlet and the exit velocity heads are negligible.

***Analysis*** The variation of fluid depth $h$ in the tank with time $t$ was determined in the previous problem to be

$$t = \frac{32\nu LD^2}{gd^4}\ln(H/h)$$

Solving for ν and substituting the given values, the kinematic viscosity of the oil is determined to be

$$\nu = \frac{gd^4}{32LD^2\ln(H/h)}t = \frac{(9.81\,\text{m/s}^2)(0.006\,\text{m})^4}{32(0.65\,\text{m})(0.63\,\text{m})^2\ln(0.4/0.36)}(2842\,\text{s}) = \mathbf{4.15\times10^{-5}\ m^2/s}$$

***Discussion*** Note that the entrance effects are not considered, and the velocity heads are disregarded. Also, the value of the viscosity strongly depends on temperature, and thus the oil temperature should be maintained constant during the test.

## Design and Essay Problems

**8-139 to 8-142**

**Solution** Students' essays and designs should be unique and will differ from each other.



# Solutions Manual for

# Fluid Mechanics: Fundamentals and Applications

# by Çengel & Cimbala

---

# CHAPTER 9

# DIFFERENTIAL ANALYSIS OF FLUID FLOW

---

## General and Mathematical Problems

### 9-1C

**Solution** We are to explain the fundamental differences between a flow domain and a control volume.

***Analysis*** **A *control volume* is used in an integral, control volume solution**. It is a volume over which all mass flow rates, forces, etc. are specified over the entire control surface of the control volume. In a control volume analysis we do not know or care about details *inside* the control volume. Rather, we solve for gross features of the flow such as net force acting on a body. **A *flow domain*, on the other hand, is also a volume, but is used in a differential analysis**. Differential equations of motion are solved everywhere inside the flow domain, and we *are* interested in all the details inside the flow domain.

***Discussion*** Note that we also need to specify what is happening at the boundaries of a flow domain – these are called *boundary conditions*.

### 9-2C

**Solution** We are to explain what we mean by coupled differential equations.

***Analysis*** A set of coupled differential equations simply means that **the equations are dependent on each other and must be solved together rather than separately**. For example, the equations of motion for fluid flow involve velocity variables in both the conservation of mass equation and the momentum equation. To solve for these variables, we must solve the coupled set of differential equations together.

***Discussion*** In some very simple fluid flow problems, the equations become uncoupled, and are easier to solve.

### 9-3C

**Solution** We are to discuss the number of unknowns and the equations needed to solve for those unknowns for a three-dimensional, unsteady, incompressible flow field.

***Analysis*** There are **four unknowns** (velocity components $u$, $v$, $w$, and pressure $P$) and thus we need to solve **four equations**:

- one from conservation of mass which is a scalar equation
- three from Newton's second law which is a vector equation

***Discussion*** These equations are also coupled in general.

### 9-4C

**Solution** We are to discuss the number of unknowns and the equations needed to solve for those unknowns for a three-dimensional, unsteady, compressible flow field with significant variations in both temperature and density.

***Analysis*** There are **six unknowns** (velocity components $u$, $v$, $w$, $\rho$, $T$, and $P$) and thus we need to solve **six equations**:

- one from conservation of mass which is a scalar equation
- three from Newton's second law which is a vector equation
- one from the energy equation which is a scalar equation
- one from an equation of state (e.g. ideal gas law) which is a scalar equation

***Discussion*** These equations are also coupled in general.

**9-5C**

**Solution** We are to express the divergence theorem in words.

***Analysis*** For vector $\vec{G}$, **the volume integral of the divergence of $\vec{G}$ over volume *V* is equal to the surface integral of the normal component of $\vec{G}$ taken over the surface *A* that encloses the volume**.

***Discussion*** The divergence theorem is also called *Gauss's theorem*.

---

**9-6**

**Solution** We are to transform a position from Cartesian to cylindrical coordinates.

***Analysis*** We use the coordinate transformations provided in this chapter,

$$r = \sqrt{x^2 + y^2} = \sqrt{(4\text{ m})^2 + (3\text{ m})^2} = 5\text{ m} \qquad (1)$$

and

$$\theta = \tan^{-1}\frac{y}{x} = \tan^{-1}\left(\frac{3\text{ m}}{4\text{ m}}\right) = 36.87^{\text{o}} = 0.6435\text{ radians} \qquad (2)$$

Coordinate *z* remains unchanged. Thus,

*Position in cylindrical coordinates*: $\vec{x} = (r, \theta, z) = (\mathbf{5\ m, 0.6435\ radians, -4\ m})$ (3)

***Discussion*** Notice that the units of $\theta$ are radians since angles are dimensionless.

---

**9-7**

**Solution** We are to calculate a truncated Taylor series expansion for a given function and compare our result with the exact value.

***Analysis*** The algebra here is simple since $d(e^x)/dx = e^x$. The Taylor series expansion is

*Taylor series expansion*: $$f(x_0 + dx) = e^{x_0} + e^{x_0}dx + \frac{1}{2}e^{x_0}dx^2 + \frac{1}{3\times 2}e^{x_0}dx^3 + ... \qquad (1)$$

We plug $x_0$ = 0 and $dx$ = –0.1 into Eq. 1,

*Truncated Taylor series expansion*:

$$f(-0.1) \approx 1 + 1\times(-0.1) + \frac{1}{2}\times 1\times(-0.1)^2 + \frac{1}{6}\times 1\times(-0.1)^3 = 0.9048333... \qquad (2)$$

We compare Eq. 2 with the exact value,

*Exact value*: $$f(-0.1) = e^{-0.1} = 0.904837418... \qquad (3)$$

Comparing Eqs. 2 and 3 we see that **our approximation is good to four or five significant digits**.

***Discussion*** The smaller the value of *dx*, the better the approximation. You can easily convince yourself of this by trying *dx* = 0.01 instead.

---

**9-8**

**Solution** We are to calculate the divergence of a given vector.

***Analysis*** The divergence of $\vec{G}$ is the dot product of the del operator $\vec{\nabla} = \frac{\partial}{\partial x}\vec{i} + \frac{\partial}{\partial y}\vec{j} + \frac{\partial}{\partial z}\vec{k}$ with $\vec{G}$, which gives

*Divergence of* $\vec{G}$:
$$\vec{\nabla}\cdot\vec{G} = \left(\frac{\partial}{\partial x}\vec{i} + \frac{\partial}{\partial y}\vec{j} + \frac{\partial}{\partial z}\vec{k}\right)\cdot\left(2xz\vec{i} - \frac{1}{2}x^2\vec{j} + z^2\vec{k}\right) = 2z + 0 - 2z = \mathbf{0}$$

**It turns out that for this special case, the divergence of $\vec{G}$ is zero**.

***Discussion*** If $\vec{G}$ were a velocity vector, this would mean that the flow field is incompressible.

---

**9-9**

**Solution** We are to perform both integrals of the divergence theorem for a given vector and volume, and verify that they are equal.

***Analysis*** We do the volume integral first:

*Volume integral*:
$$\int_V \vec{\nabla}\cdot\vec{G}dV = \int_{x=0}^{x=1}\int_{y=0}^{y=1}\int_{z=0}^{z=1}\left(\frac{\partial G_x}{\partial x} + \frac{\partial G_y}{\partial y} + \frac{\partial G_z}{\partial z}\right)dzdydx = \int_{x=0}^{x=1}\int_{y=0}^{y=1}\int_{z=0}^{z=1}\left(4z - 2y + y\right)dzdydx \qquad (1)$$

The term in parentheses in Eq. 1 reduces to $(4z - y)$, and we integrate this over $z$ first,

$$\int_V \vec{\nabla}\cdot\vec{G}dV = \int_{x=0}^{x=1}\int_{y=0}^{y=1}\left[2z^2 - yz\right]_{z=0}^{z=1} dydx = \int_{x=0}^{x=1}\int_{y=0}^{y=1}\left(2 - y\right)dydx$$

Then we integrate over $y$ and then over $x$,

*Volume integral*:
$$\int_V \vec{\nabla}\cdot\vec{G}dV = \int_{x=0}^{x=1}\left[2y - \frac{y^2}{2}\right]_{y=0}^{y=1} dx = \int_{x=0}^{x=1}\frac{3}{2}dx = \mathbf{\frac{3}{2}} \qquad (2)$$

Next we calculate the surface integral of the divergence theorem. There are six faces of the cube, and unit vector $\vec{n}$ points outward from each face. So, we split the area integral into six parts and sum them. E.g., the right-most face has $\vec{n}$ = (1,0,0), so $\vec{G}\cdot\vec{n}$ = $4xz$ on this face. The bottom face has $\vec{n}$ = (0,–1,0), so $\vec{G}\cdot\vec{n}$ = $y^2$ on this face. The surface integral is then

*Surface integral*:
$$\oint_A \vec{G}\cdot\vec{n}dA = \underbrace{\left[\int_{y=0}^{y=1}\int_{z=0}^{z=1}\left(4xz\right)dzdy\right]_{x=1}}_{\text{Right face}} + \underbrace{\left[\int_{y=0}^{y=1}\int_{z=0}^{z=1}\left(-4xz\right)dzdy\right]_{x=0}}_{\text{Left face}} + \underbrace{\left[\int_{z=0}^{z=1}\int_{x=0}^{x=1}\left(-y^2\right)dxdz\right]_{y=1}}_{\text{Top face}}$$
$$+ \underbrace{\left[\int_{z=0}^{z=1}\int_{x=0}^{x=1}\left(y^2\right)dxdz\right]_{y=0}}_{\text{Bottom face}} + \underbrace{\left[\int_{x=0}^{x=1}\int_{y=0}^{y=1}\left(yz\right)dydx\right]_{z=1}}_{\text{Front face}} + \underbrace{\left[\int_{x=0}^{x=1}\int_{y=0}^{y=1}\left(-yz\right)dydx\right]_{z=0}}_{\text{Back face}} \qquad (3)$$

The three integrals on the far right of Eq. 3 are obviously zero. The other three integrals can be obtained carefully,

$$\oint_A \vec{G}\cdot\vec{n}dA = \int_{y=0}^{y=1}\left[2z^2\right]_{z=0}^{z=1}dy + \int_{z=0}^{z=1}\left[-x\right]_{x=0}^{x=1}dz + \int_{x=0}^{x=1}\left[\frac{y^2}{2}\right]_{y=0}^{y=1}dx = \int_{y=0}^{y=1}\left(2\right)dy + \int_{z=0}^{z=1}\left(-1\right)dz + \int_{x=0}^{x=1}\left(\frac{1}{2}\right)dx \qquad (4)$$

The last three integrals of Eq. 4 are trivial. The final result is

*Surface integral*:
$$\oint_A \vec{G}\cdot\vec{n}dA = 2 - 1 + \frac{1}{2} = \mathbf{\frac{3}{2}} \qquad (5)$$

**Since Eq. 2 and Eq. 5 are equal, the divergence theorem works for this case**.

***Discussion*** The integration is simple in this example since each face is flat and normal to an axis. In the general case in which the surface is curved, integration is much more difficult, but the divergence theorem always works.

---

**9-10**

**Solution** We are to expand a dot product in Cartesian coordinates and verify it.

***Analysis*** In Cartesian coordinates the del operator is $\vec{\nabla} = \frac{\partial}{\partial x}\vec{i} + \frac{\partial}{\partial y}\vec{j} + \frac{\partial}{\partial z}\vec{k}$ and we let $\vec{G} = G_x\vec{i} + G_y\vec{j} + G_z\vec{k}$. The left hand side of the equation is thus

*Left hand side*:

$$\vec{\nabla}\cdot\left(f\vec{G}\right) = \frac{\partial\left(fG_x\right)}{\partial x} + \frac{\partial\left(fG_y\right)}{\partial y} + \frac{\partial\left(fG_z\right)}{\partial z}$$
$$= G_x\frac{\partial f}{\partial x} + f\frac{\partial G_x}{\partial x} + G_y\frac{\partial f}{\partial y} + f\frac{\partial G_y}{\partial y} + G_z\frac{\partial f}{\partial z} + f\frac{\partial G_z}{\partial z} \quad (1)$$

The right hand side of the equation is

*Right hand side*:

$$\vec{G}\cdot\vec{\nabla}f + f\vec{\nabla}\cdot\vec{G}$$
$$= \left(G_x\vec{i} + G_y\vec{j} + G_z\vec{k}\right)\cdot\left(\frac{\partial f}{\partial x}\vec{i} + \frac{\partial f}{\partial y}\vec{j} + \frac{\partial f}{\partial z}\vec{k}\right) + f\left(\frac{\partial G_x}{\partial x} + \frac{\partial G_y}{\partial y} + \frac{\partial G_z}{\partial z}\right) \quad (2)$$
$$= G_x\frac{\partial f}{\partial x} + G_y\frac{\partial f}{\partial y} + G_z\frac{\partial f}{\partial z} + f\frac{\partial G_x}{\partial x} + f\frac{\partial G_y}{\partial y} + f\frac{\partial G_z}{\partial z}$$

Equations 1 and 2 are the same, and **the given equation is verified**.

***Discussion*** The product rule given in this problem was used in this chapter in the derivation of the alternative form of the continuity equation.

**9-11**

**Solution** We are to expand the given equation in Cartesian coordinates and verify it.

***Analysis*** In Cartesian coordinates the del operator is $\vec{\nabla} = \frac{\partial}{\partial x}\vec{i} + \frac{\partial}{\partial y}\vec{j} + \frac{\partial}{\partial z}\vec{k}$ and we let $\vec{F} = F_x\vec{i} + F_y\vec{j} + F_z\vec{k}$ and $\vec{G} = G_x\vec{i} + G_y\vec{j} + G_z\vec{k}$. The left hand side of the equation is thus

$$\vec{\nabla}\cdot\left(\vec{F}\vec{G}\right) = \begin{bmatrix} \frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \end{bmatrix} \begin{bmatrix} F_xG_x & F_xG_y & F_xG_z \\ F_yG_x & F_yG_y & F_yG_z \\ F_zG_x & F_zG_y & F_zG_z \end{bmatrix}$$

$$= \left[\frac{\partial}{\partial x}\left(F_xG_x\right) + \frac{\partial}{\partial y}\left(F_yG_x\right) + \frac{\partial}{\partial z}\left(F_zG_x\right)\right]\vec{i} + \left[\frac{\partial}{\partial x}\left(F_xG_y\right) + \frac{\partial}{\partial y}\left(F_yG_y\right) + \frac{\partial}{\partial z}\left(F_zG_y\right)\right]\vec{j} + \left[\frac{\partial}{\partial x}\left(F_xG_z\right) + \frac{\partial}{\partial y}\left(F_yG_z\right) + \frac{\partial}{\partial z}\left(F_zG_z\right)\right]\vec{k} \quad (1)$$

We use the product rule on each term in Eq. 1 and rearrange to get

*Left hand side*:

$$\vec{\nabla}\cdot\left(\vec{F}\vec{G}\right) = \left[G_x\left(\frac{\partial F_x}{\partial x} + \frac{\partial F_y}{\partial y} + \frac{\partial F_z}{\partial z}\right) + \left(F_x\frac{\partial}{\partial x} + F_y\frac{\partial}{\partial y} + F_z\frac{\partial}{\partial z}\right)G_x\right]\vec{i} + \left[G_y\left(\frac{\partial F_x}{\partial x} + \frac{\partial F_y}{\partial y} + \frac{\partial F_z}{\partial z}\right) + \left(F_x\frac{\partial}{\partial x} + F_y\frac{\partial}{\partial y} + F_z\frac{\partial}{\partial z}\right)G_y\right]\vec{j} + \left[G_z\left(\frac{\partial F_x}{\partial x} + \frac{\partial F_y}{\partial y} + \frac{\partial F_z}{\partial z}\right) + \left(F_x\frac{\partial}{\partial x} + F_y\frac{\partial}{\partial y} + F_z\frac{\partial}{\partial z}\right)G_z\right]\vec{k} \quad (2)$$

We recognize that $\frac{\partial F_x}{\partial x} + \frac{\partial F_y}{\partial y} + \frac{\partial F_z}{\partial z} = \vec{\nabla}\cdot\vec{F}$ and $F_x\frac{\partial}{\partial x} + F_y\frac{\partial}{\partial y} + F_z\frac{\partial}{\partial z} = \vec{F}\cdot\vec{\nabla}$. Eq. 2 then becomes

*Left hand side*:

$$\vec{\nabla}\cdot\left(\vec{F}\vec{G}\right) = \left[G_x\left(\vec{\nabla}\cdot\vec{F}\right) + \left(\vec{F}\cdot\vec{\nabla}\right)G_x\right]\vec{i} + \left[G_y\left(\vec{\nabla}\cdot\vec{F}\right) + \left(\vec{F}\cdot\vec{\nabla}\right)G_y\right]\vec{j} + \left[G_z\left(\vec{\nabla}\cdot\vec{F}\right) + \left(\vec{F}\cdot\vec{\nabla}\right)G_z\right]\vec{k} \quad (3)$$

After rearrangement, Eq. 3 becomes

*Left hand side*:

$$\vec{\nabla}\cdot\left(\vec{F}\vec{G}\right) = \left(G_x\vec{i} + G_y\vec{j} + G_z\vec{k}\right)\left(\vec{\nabla}\cdot\vec{F}\right) + \left(\vec{F}\cdot\vec{\nabla}\right)\left(G_x\vec{i} + G_y\vec{j} + G_z\vec{k}\right) \quad (4)$$

Finally, recognizing vector $\vec{G}$ twice in Eq. 4, we see that the left hand side of the given equation is identical to the right hand side, and **the given equation is verified**.

***Discussion*** It may seem surprising, but $\vec{F}\vec{G} \neq \vec{G}\vec{F}$.

**9-12**

**Solution** We are to prove the equation.

***Analysis*** We let $\vec{F} = \rho\vec{V}$ and $\vec{G} = \vec{V}$. Using Eq. 1 of the previous problem, we have

$$\vec{\nabla}\cdot\left(\rho\vec{V}\vec{V}\right) = \vec{V}\vec{\nabla}\cdot\left(\rho\vec{V}\right) + \left(\rho\vec{V}\cdot\vec{\nabla}\right)\vec{V} \qquad (1)$$

However, since the density is not operated on in the second term of Eq. 1, it can be brought outside of the parenthesis, even though it is not a constant in general. Equation 1 can thus be written as

$$\boxed{\vec{\nabla}\cdot\left(\rho\vec{V}\vec{V}\right) = \vec{V}\vec{\nabla}\cdot\left(\rho\vec{V}\right) + \rho\left(\vec{V}\cdot\vec{\nabla}\right)\vec{V}} \qquad (2)$$

***Discussion*** Equation 2 was used in this chapter in the derivation of the alternative form of Cauchy's equation.

**9-13**

**Solution** We are to transform cylindrical velocity components to Cartesian velocity components.

***Analysis*** We apply trigonometry, recognizing that the angle between $u$ and $u_r$ is $\theta$, and the angle between $v$ and $u_\theta$ is also $\theta$,

*x component of velocity*: $\boxed{u = u_r\cos\theta - u_\theta\sin\theta}$ (1)

Similarly,

*y component of velocity*: $\boxed{v = u_r\sin\theta + u_\theta\cos\theta}$ (2)

The transformation of the *z* component is trivial,

*z component of velocity*: $\boxed{w = u_z}$ (3)

***Discussion*** These transformations come in handy.

**9-14**

**Solution** We are to transform Cartesian velocity components to cylindrical velocity components.

***Analysis*** We apply trigonometry, recognizing that the angle between $u$ and $u_r$ is $\theta$, and the angle between $v$ and $u_\theta$ is also $\theta$,

*$u_r$ component of velocity*: $\boxed{u_r = u\cos\theta + v\sin\theta}$ (1)

Similarly,

*$u_\theta$ component of velocity*:: $\boxed{u_\theta = -u\sin\theta + v\cos\theta}$ (2)

The transformation of the *z* component is trivial,

*z component of velocity*: $\boxed{u_z = w}$ (3)

***Discussion*** You can also obtain Eqs. 1 and 2 by solving Eqs. 1 and 2 of the previous problem simultaneously.

## 9-15

**Solution** We are to transform a given set of Cartesian coordinates and velocity components into cylindrical coordinates and velocity components.

***Analysis*** First we apply the coordinate transformations given in this chapter,

$$r = \sqrt{x^2 + y^2} = \sqrt{\left(0.50\text{ m}\right)^2 + \left(0.20\text{ m}\right)^2} = 0.5385\text{ m} \qquad (1)$$

$$\theta = \tan^{-1}\frac{y}{x} = \tan^{-1}\left(\frac{0.20\text{ m}}{0.50\text{ m}}\right) = 21.80^{\circ} = 0.3805\text{ radians} \qquad (2)$$

Next we apply the results of the previous problem,

$$u_r = u\cos\theta + v\sin\theta = 10.3\frac{\text{m}}{\text{s}}\times\frac{0.50\text{ m}}{0.5385\text{ m}} - 5.6\frac{\text{m}}{\text{s}}\times\frac{0.20\text{ m}}{0.5385\text{ m}} = 7.484\frac{\text{m}}{\text{s}} \qquad (3)$$

$$u_\theta = -u\sin\theta + v\cos\theta = -10.3\frac{\text{m}}{\text{s}}\times\frac{0.20\text{ m}}{0.5385\text{ m}} - 5.6\frac{\text{m}}{\text{s}}\times\frac{0.50\text{ m}}{0.5385\text{ m}} = -9.025\frac{\text{m}}{\text{s}} \qquad (4)$$

Note that we have used the fact that $x = r\cos\theta$ and $y = r\sin\theta$ for convenience in Eqs. 3 and 4. Our final results are summarized to three significant digits:

*Results*:

$$\boxed{r = 0.539\text{ m},\ \ \theta = 0.381\text{ radians},\ \ u_r = 7.48\frac{\text{m}}{\text{s}},\ \ u_\theta = -9.03\frac{\text{m}}{\text{s}}} \qquad (5)$$

We verify our result by calculating the square of the speed in both coordinate systems. In Cartesian coordinates,

$$V^2 = u^2 + v^2 = \left(10.3\frac{\text{m}}{\text{s}}\right)^2 + \left(-5.6\frac{\text{m}}{\text{s}}\right)^2 = 137.5\frac{\text{m}^2}{\text{s}^2} \qquad (6)$$

In cylindrical coordinates,

$$V^2 = u_r^{\ 2} + u_\theta^{\ 2} = \left(7.484\frac{\text{m}}{\text{s}}\right)^2 + \left(-9.025\frac{\text{m}}{\text{s}}\right)^2 = 137.5\frac{\text{m}^2}{\text{s}^2} \qquad (7)$$

***Discussion*** Such checks of our algebra are always wise.

---

## 9-16

**Solution** We are to transform a given set of Cartesian velocity components into cylindrical velocity components, and identify the flow.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* or *r*-$\theta$ plane.

***Analysis*** We recognize that $r^2 = x^2 + y^2$. We also know that $y = r\sin\theta$ and $x = r\cos\theta$. Using the results of Problem 9-14, the cylindrical velocity components are

$u_r$ *component of velocity*:

$$\boxed{u_r = u\cos\theta + v\sin\theta = \frac{Cr\sin\theta\cos\theta}{r^2} - \frac{Cr\sin\theta\cos\theta}{r^2} = 0} \qquad (1)$$

$u_\theta$ *component of velocity*::

$$\boxed{u_\theta = -u\sin\theta + v\cos\theta = -\frac{Cr\sin^2\theta}{r^2} - \frac{Cr\cos^2\theta}{r^2} = \frac{-C}{r}} \qquad (2)$$

where we have also used the fact that $\cos^2\theta + \sin^2\theta = 1$. We recognize the velocity components of Eqs. 1 and 2 as those of a **line vortex**.

***Discussion*** The negative sign in Eq. 2 indicates that this vortex is in the *clockwise* direction.

---

**9-17**

**Solution** We are to transform a given set of cylindrical velocity components into Cartesian velocity components.

***Analysis*** We apply the coordinate transformations given in this chapter, along with the results of Problem 9-13,

*x component of velocity*: $$u = u_r \cos\theta - u_\theta \sin\theta = \frac{m}{2\pi r}\frac{x}{r} - \frac{\Gamma}{2\pi r}\frac{y}{r} \quad (1)$$

We recognize that $r^2 = x^2 + y^2$. Thus, Eq. 1 becomes

*x component of velocity*: $$u = \frac{1}{2\pi\left(x^2+y^2\right)}\left(mx - \Gamma y\right) \quad (2)$$

Similarly,

*y component of velocity*: $$v = u_r \sin\theta + u_\theta \cos\theta = \frac{m}{2\pi r}\frac{y}{r} + \frac{\Gamma}{2\pi r}\frac{x}{r} \quad (3)$$

Again recognizing that $r^2 = x^2 + y^2$, Eq. 3 becomes

*y component of velocity*: $$v = \frac{1}{2\pi\left(x^2+y^2\right)}\left(my + \Gamma x\right) \quad (4)$$

We verify our result by calculating the square of the speed in both coordinate systems. In Cartesian coordinates,

$$V^2 = u^2 + v^2 = \frac{1}{4\pi^2\left(x^2+y^2\right)^2}\left(m^2x^2 - 2mx\Gamma y + \Gamma^2 y^2\right) + \frac{1}{4\pi^2\left(x^2+y^2\right)^2}\left(m^2y^2 + 2my\Gamma x + \Gamma^2 x^2\right) \quad (5)$$

Two of the terms in Eq. 5 cancel, and we combine the others. After simplification,

*Magnitude of velocity squared*: $$V^2 = u^2 + v^2 = \frac{1}{4\pi^2\left(x^2+y^2\right)}\left(m^2 + \Gamma^2\right) \quad (6)$$

We calculate $V^2$ from the components given in cylindrical coordinates as well,

*Magnitude of velocity squared*: $$V^2 = {u_r}^2 + {u_\theta}^2 = \frac{m^2}{4\pi^2 r^2} + \frac{\Gamma^2}{4\pi^2 r^2} = \frac{m^2 + \Gamma^2}{4\pi^2 r^2} \quad (7)$$

Finally, since $r^2 = x^2 + y^2$, Eqs. 6 and 7 are the same, and **the results are verified**.

***Discussion*** Such checks of our algebra are always wise.

## 9-18E

**Solution** We are to transform a given set of Cartesian coordinates and velocity components into cylindrical coordinates and velocity components.

***Analysis*** First we apply the coordinate transformations given in this chapter,

$$x = r\cos\theta = 6.20\text{ in}\times\cos\left(30.0^{\circ}\right)\left(\frac{1\text{ ft}}{12\text{ in}}\right) = 0.4474\text{ ft} \qquad (1)$$

and

$$y = r\sin\theta = 6.20\text{ in}\times\sin\left(30.0^{\circ}\right)\left(\frac{1\text{ ft}}{12\text{ in}}\right) = 0.2583\text{ ft} \qquad (2)$$

Next we apply the results of Problem 9-13,

$$u = u_r\cos\theta - u_\theta\sin\theta = 1.37\frac{\text{ft}}{\text{s}}\times\cos\left(30.0^{\circ}\right) - 3.82\frac{\text{ft}}{\text{s}}\times\sin\left(30.0^{\circ}\right) = -0.7235\frac{\text{ft}}{\text{s}} \qquad (3)$$

and

$$v = u_r\sin\theta + u_\theta\cos\theta = 1.37\frac{\text{ft}}{\text{s}}\times\sin\left(30.0^{\circ}\right) + 3.82\frac{\text{ft}}{\text{s}}\times\cos\left(30.0^{\circ}\right) = 3.993\frac{\text{ft}}{\text{s}} \qquad (4)$$

Our final results are summarized to three significant digits:

*Results*:

$$x = \mathbf{0.447\ ft},\ \ y = \mathbf{0.258\ ft},\ \ u = \mathbf{-0.724\frac{ft}{s}},\ \ v = \mathbf{3.99\frac{ft}{s}} \qquad (5)$$

We verify our result by calculating the square of the speed in both coordinate systems. In Cartesian coordinates,

$$V^2 = u^2 + v^2 = \left(-0.7235\frac{\text{ft}}{\text{s}}\right)^2 + \left(3.993\frac{\text{ft}}{\text{s}}\right)^2 = \mathbf{16.47\frac{ft^2}{s^2}} \qquad (6)$$

In cylindrical coordinates,

$$V^2 = u_r^{\ 2} + u_\theta^{\ 2} = \left(1.37\frac{\text{ft}}{\text{s}}\right)^2 + \left(3.82\frac{\text{ft}}{\text{s}}\right)^2 = \mathbf{16.47\frac{ft^2}{s^2}} \qquad (7)$$

***Discussion*** Such checks of our algebra are always wise.

## Continuity Equation

### 9-19C

**Solution** We are to discuss the material derivative of density for the case of compressible and incompressible flow.

***Analysis*** If the flow field is compressible, we expect that as a fluid particle (a *material element*) moves around in the flow, its density changes. Thus the *material derivative of density* (the rate of change of density following a fluid particle) is **non-zero for compressible flow**. However, if the flow field is incompressible, the density remains constant. As a fluid particle moves around in the flow, the material derivative of density must be **zero for incompressible flow** (no change in density following the fluid particle).

***Discussion*** The material derivative of any property is the rate of change of that property following a fluid particle.

### 9-20C

**Solution** We are to explain why the derivation of the continuity via the divergence theorem is so much less involved than the derivation of the same equation by summation of mass flow rates through each face of an infinitesimal control volume.

***Analysis*** In the derivation using the divergence theorem, we begin with the control volume form of conservation of mass, and simply apply the divergence theorem. The control volume form was already derived in Chap. 5, so **we begin the derivation in this chapter with an established conservation of mass equation**. On the other hand, the alternative derivation is from "scratch" and therefore requires much more algebra.

***Discussion*** The bottom line is that the divergence theorem enables us to quickly convert the control volume form of the conservation law into the differential form.

**9-21**

**Solution** For given velocity component $u$ and density $\rho$, we are to predict velocity component $v$, plot an approximate shape of the duct, and predict its height at section (2).

***Assumptions*** **1** The flow is steady and two-dimensional in the $x$-$y$ plane, but *compressible*. **2** Friction on the walls is ignored. **3** Axial velocity $u$ and density $\rho$ vary linearly with $x$. **4** The $x$ axis is a line of top-bottom symmetry.

***Properties*** The fluid is standard air. The speed of sound is about 340 m/s, so the flow is subsonic, but compressible.

***Analysis*** ($a$) We write expressions for $u$ and $\rho$, forcing them to be linear in $x$,

$$u = u_1 + C_u x \qquad C_u = \frac{u_2 - u_1}{\Delta x} = \frac{(100-300)\,\frac{\text{m}}{\text{s}}}{2.0\text{ m}} = -100\,\frac{1}{\text{s}} \tag{1}$$

$$\rho = \rho_1 + C_\rho x \qquad C_\rho = \frac{\rho_2 - \rho_1}{\Delta x} = \frac{(1.2-0.85)\,\frac{\text{kg}}{\text{m}^3}}{2.0\text{ m}} = 0.175\,\frac{\text{kg}}{\text{m}^4} \tag{2}$$

where $C_u$ and $C_\rho$ are constants. We use the compressible form of the steady continuity equation, placing the unknown term $v$ on the left hand side, and plugging in Eqs. 1 and 2,

$$\frac{\partial(\rho v)}{\partial y} = -\frac{\partial(\rho u)}{\partial x} = -\frac{\partial\left((\rho_1 + C_\rho x)(u_1 + C_u x)\right)}{\partial x}$$

After some algebra,

$$\frac{\partial(\rho v)}{\partial y} = -\left(\rho_1 C_u + u_1 C_\rho\right) - 2C_u C_\rho x \tag{3}$$

We integrate Eq. 3 with respect to $y$,

$$\rho v = -\left(\rho_1 C_u + u_1 C_\rho\right) y - 2C_u C_\rho xy + f(x) \tag{4}$$

Since this is a *partial* integration, we add an arbitrary function of $x$ instead of simply a constant of integration. We now apply boundary conditions. Since the flow is symmetric about the $x$ axis ($y$ = 0), $v$ must equal zero at $y$ = 0 for any $x$. This is possible only if $f(x)$ is identically zero. Applying $f(x)$ = 0, dividing by $\rho$ to solve for $v$, and plugging in Eq. 2, Eq. 4 becomes

$$\boxed{v = \frac{-\left(\rho_1 C_u + u_1 C_\rho\right) y - 2C_u C_\rho xy}{\rho} = \frac{-\left(\rho_1 C_u + u_1 C_\rho\right) y - 2C_u C_\rho xy}{\rho_1 + C_\rho x}} \tag{5}$$

($b$) For known values of $u$ and $v$, we can plot streamlines between $x$ = 0 and $x$ = 2.0 m using the technique described in Chap. 4. Several streamlines are shown in Fig. 1. The streamline starting at $x$ = 0, $y$ = 0.8 m is the top wall of the duct.

($c$) At section (2), the top streamline crosses $y$ = 1.70 m at $x$ = 2.0 m. Thus, *the predicted height of the duct at section* (2) *is* 1.70 m.

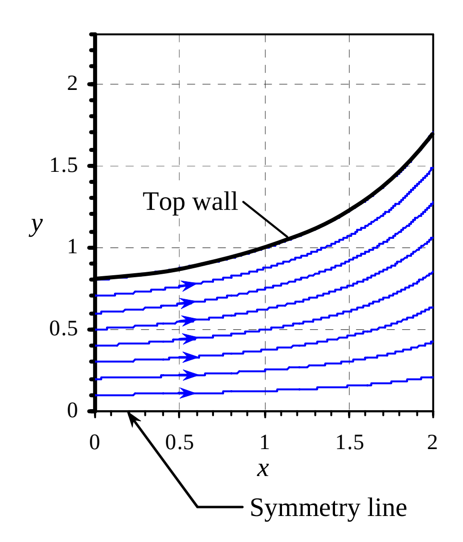



**FIGURE 1**
Streamlines for a diverging duct.

***Discussion*** You can verify that the combination of Eqs. 1, 2, and 5 satisfies the steady compressible continuity equation. However, this alone does not guarantee that the density and velocity components will actually *follow* these equations if this diverging duct were to be built. The actual flow depends on the *pressure rise* between sections (1) and (2) – only one unique pressure rise can yield the desired flow deceleration. Temperature may also change considerably in this kind of compressible flow field.

## 9-22

**Solution** We are to repeat Example 9-1, but without using continuity.

***Assumptions*** **1** Density varies with time, but not space; in other words, the density is uniform throughout the cylinder at any given time, but changes with time. **2** No mass escapes from the cylinder during the compression.

***Analysis*** The mass inside the cylinder is constant, but the volume decreases linearly as the piston moves up. At $t = 0$ when $L = L_{Bottom}$ the initial volume of the cylinder is $V(0) = L_{Bottom}A$, where $A$ is the cross-sectional area of the cylinder. At $t = 0$ the density is $\rho = \rho(0) = m/V(0)$, and thus

*Mass in the cylinder*:
$$m = \rho(0)V(0) = \rho(0)L_{Bottom}A \qquad (1)$$

Mass $m$ (Eq. 1) is a constant since no mass escapes during the compression. At some later time $t$, $L = L_{Bottom} - V_P t$ and the volume is thus

*Cylinder volume at time t*:
$$V = (L_{Bottom} - V_P t)A \qquad (2)$$

The density at time $t$ is

*Density at time t*:
$$\rho = \frac{m}{V} = \frac{\rho(0)L_{Bottom}A}{(L_{Bottom} - V_P t)A} \qquad (3)$$

where we have plugged in Eq. 1 for $m$ and Eq. 2 for $V$. Equation 3 reduces to

$$\rho = \rho(0)\frac{L_{Bottom}}{L_{Bottom} - V_P t} \qquad (4)$$

or, using the nondimensional variables of Example 9-1,

*Nondimensional result*:
$$\boxed{\frac{\rho}{\rho(0)} = \frac{1}{1 - \frac{V_P t}{L_{Bottom}}} \qquad \text{or} \qquad \rho^* = \frac{1}{1 - t^*}} \qquad (5)$$

which is identical to Eq. 5 of Example 9-1.

***Discussion*** We see by this exercise that the continuity equation is indeed an equation of conservation of mass.

---

## 9-23

**Solution** We are to expand the continuity equation in Cartesian coordinates.

***Analysis*** We expand the second term by taking the dot product of the del operator $\vec{\nabla} = \left(\frac{\partial}{\partial x}\vec{i} + \frac{\partial}{\partial y}\vec{j} + \frac{\partial}{\partial z}\vec{k}\right)$ with $\rho\vec{V} = (\rho u)\vec{i} + (\rho v)\vec{j} + (\rho w)\vec{k}$, giving

*Compressible continuity equation in Cartesian coordinates:*
$$\boxed{\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0} \qquad (1)$$

We can further expand Eq. 1 by using the product rule on the spatial derivatives, resulting in 7 terms,

*Further expansion:*
$$\boxed{\frac{\partial \rho}{\partial t} + \rho\frac{\partial u}{\partial x} + u\frac{\partial \rho}{\partial x} + \rho\frac{\partial v}{\partial y} + v\frac{\partial \rho}{\partial y} + \rho\frac{\partial w}{\partial z} + w\frac{\partial \rho}{\partial z} = 0} \qquad (2)$$

***Discussion*** We can do a similar thing in cylindrical coordinates, but the algebra is somewhat more complicated.

---

## 9-24

**Solution** We are to write the given equation as a word equation and discuss it.

***Analysis*** Here is a word equation: "**The time rate of change of volume of a fluid particle per unit volume is equal to the divergence of the velocity field**." As a fluid particle moves around in a compressible flow, it can distort, rotate, and get larger or smaller. Thus the volume of the fluid element can change with time; this is represented by the left hand side of the equation. The right hand side is identically zero for an incompressible flow, but it is *not* zero for a compressible flow. Thus we can think of the volumetric strain rate as a measure of compressibility of a fluid flow.

***Discussion*** Volumetric strain rate is a kinematic property as discussed in Chap. 4. Nevertheless, it is shown here to be related to the continuity equation (conservation of mass).

---

## 9-25

**Solution** We are to verify that a given flow field satisfies the continuity equation, and we are to discuss conservation of mass at the origin.

***Analysis*** The 2-D cylindrical velocity components ($u_r$,$u_\theta$) for this flow field are

*Cylindrical velocity components*:

$$u_r = \frac{m}{2\pi r} \qquad u_\theta = \frac{\Gamma}{2\pi r} \qquad (1)$$

where $m$ and $\Gamma$ are constants We plug Eq. 1 into the incompressible continuity equation in cylindrical coordinates,

*Incompressible continuity*:

$$\frac{1}{r}\frac{\partial(ru_r)}{\partial r} + \frac{1}{r}\frac{\partial(u_\theta)}{\partial\theta} + \frac{\partial(u_z)}{\partial z} = 0 \quad \text{or} \quad \frac{1}{r}\underbrace{\frac{\partial\left(\frac{m}{2\pi}\right)}{\partial r}}_{0} + \frac{1}{r}\underbrace{\frac{\partial\left(\frac{\Gamma}{2\pi r}\right)}{\partial\theta}}_{0} + \underbrace{\frac{\partial(u_z)}{\partial z}}_{0} = 0 \qquad (2)$$

The first term is zero because it is the derivative of a constant. The second term is zero because $r$ is not a function of $\theta$. The third term is zero since this is a 2-D flow with $u_z$ = 0. Thus, we verify that *the incompressible continuity equation is satisfied for the given velocity field*.

At the origin, both $u_r$ and $u_\theta$ go to infinity. Conservation of mass is not affected by $u_\theta$, but the fact that $u_r$ is non-zero at the origin violates conservation of mass. We think of the flow along the $z$ axis as a *line sink* toward which mass approaches from all directions in the plane and then disappears (like a black hole in two dimensions). *Mass is not conserved at the origin*.

***Discussion*** Singularities such as this are unphysical of course, but are nevertheless useful as approximations of real flows, as long as we stay away from the singularity itself.

---

**9-26**

**Solution** We are to verify that a given velocity field satisfies continuity.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x-y* plane.

***Analysis*** The velocity field of Problem 9-16 is

*Cartesian velocity components*:
$$u = \frac{Cy}{x^2 + y^2} \qquad v = \frac{-Cx}{x^2 + y^2} \qquad (1)$$

We check continuity, staying in Cartesian coordinates,

$$\underbrace{\frac{\partial u}{\partial x}}_{-2xCy\left(x^2+y^2\right)^{-3}} + \underbrace{\frac{\partial v}{\partial y}}_{2yCx\left(x^2+y^2\right)^{-3}} + \underbrace{\frac{\partial w}{\partial z}}_{0 \text{ since 2-D}} = 0$$

So we see that **the incompressible continuity equation is indeed satisfied**.

***Discussion*** The fact that the flow field satisfies continuity does not guarantee that a corresponding pressure field exists that can satisfy the steady conservation of momentum equation. In this case, however, it does.

---

**9-27**

**Solution** We are to verify that a given velocity field is incompressible.

***Assumptions*** **1** The flow is two-dimensional, implying no *z component* of velocity and no variation of *u* or *v* with *z*.

***Analysis*** The components of velocity in the *x* and *y* directions respectively are

$$u = 1.3 + 2.8x \qquad v = 1.5 - 2.8y$$

To check if the flow is incompressible, we see if the incompressible continuity equation is satisfied:

$$\underbrace{\frac{\partial u}{\partial x}}_{2.8} + \underbrace{\frac{\partial v}{\partial y}}_{-2.8} + \underbrace{\frac{\partial w}{\partial z}}_{0 \text{ since 2-D}} = 0 \qquad \text{or} \qquad 2.8 - 2.8 = 0$$

So we see that **the incompressible continuity equation is indeed satisfied**. Hence the flow field is incompressible.

***Discussion*** The fact that the flow field satisfies continuity does not guarantee that a corresponding pressure field exists that can satisfy the steady conservation of momentum equation.

---

**9-28**

**Solution** We are to find the most general form of the radial velocity component of a purely radial flow that does not violate conservation of mass.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* or *r*-$\theta$ plane.

***Analysis*** We use cylindrical coordinates for convenience. We solve for $u_r$ using the incompressible continuity equation,

$$\frac{1}{r}\frac{\partial(ru_r)}{\partial r}+\underbrace{\frac{1}{r}\frac{\partial(u_\theta)}{\partial\theta}}_{0\text{ for radial flow}}+\underbrace{\frac{\partial(u_z)}{\partial z}}_{0\text{ for 2-D flow}}=0 \quad\text{or}\quad \frac{\partial(ru_r)}{\partial r}=0 \qquad (1)$$

We integrate Eq. 1 with respect to *r*, adding a function of the other variable $\theta$ rather than simply a constant of integration since this is a partial integration,

*Result*:
$$\boxed{ru_r=f(\theta) \quad\text{or}\quad u_r=\frac{f(\theta)}{r}} \qquad (2)$$

***Discussion*** Any function of $\theta$ in Eq. 2 will satisfy the continuity equation.

---

**9-29**

**Solution** We are to determine a relationship between constants *a*, *b*, *c*, and *d* that ensures incompressibility.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible (under certain restraints to be determined).

***Analysis*** We plug the given velocity components into the incompressible continuity equation,

*Condition for incompressibility*:
$$\underbrace{\frac{\partial u}{\partial x}}_{ay^2}+\underbrace{\frac{\partial v}{\partial y}}_{3cy^2}+\underbrace{\frac{\partial w}{\partial z}}_{0}=0 \qquad ay^2+3cy^2=0$$

Thus to guarantee incompressibility, constants *a* and *c* must satisfy the following relationship:

*Condition for incompressibility*:
$$\boxed{a=-3c} \qquad (1)$$

***Discussion*** If Eq. 1 were not satisfied, the given velocity field might still represent a valid flow field, but density would have to vary with location in the flow field – in other words the flow would be *compressible*.

---

**9-30**

**Solution** We are to find the *y component* of velocity, *v*, using a given expression for *u*.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane, implying that *w* = 0 and neither *u* nor *v* depend on *z*.

***Analysis*** Since the flow is steady and incompressible, we apply the incompressible continuity in Cartesian coordinates to the flow field, giving

*Condition for incompressibility*:
$$\frac{\partial v}{\partial y} = -\underbrace{\frac{\partial u}{\partial x}}_{a} - \underbrace{\frac{\partial w}{\partial z}}_{0} \qquad \frac{\partial v}{\partial y} = -a$$

Next we integrate with respect to *y*. Note that since the integration is a *partial* integration, we must add some arbitrary function of *x* instead of simply a constant of integration.

*Solution*:
$$\boxed{v = -ay + f(x)}$$

If the flow were three-dimensional, we would add a function of *x* and *z* instead.

***Discussion*** To satisfy the incompressible continuity equation, any function of *x* will work since there are no derivatives of *v* with respect to *x* in the continuity equation. Not all functions of *x* are necessarily physically possible, however, since the flow must also satisfy the steady conservation of momentum equation.

**9-31**

**Solution** We are to find the most general form of the tangential velocity component of a purely circular flow that does not violate conservation of mass.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* or *r*-$\theta$ plane.

***Analysis*** We use cylindrical coordinates for convenience. We solve for $u_\theta$ using the incompressible continuity equation,

$$\underbrace{\frac{1}{r}\frac{\partial(ru_r)}{\partial r}}_{\text{0 for circular flow}} + \frac{1}{r}\frac{\partial(u_\theta)}{\partial\theta} + \underbrace{\frac{\partial(u_z)}{\partial z}}_{\text{0 for 2-D flow}} = 0 \quad \text{or} \quad \frac{\partial(u_\theta)}{\partial\theta} = 0 \qquad (1)$$

We integrate Eq. 1 with respect to $\theta$, adding a function of the other variable *r* rather than simply a constant of integration since this is a partial integration,

*Result*:
$$\boxed{u_\theta = f(r)} \qquad (2)$$

***Discussion*** Any function of *r* in Eq. 2 will satisfy the continuity equation.

## 9-32

**Solution** We are to find the *y component* of velocity, *v*, using a given expression for *u*.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane, implying that *w* = 0 and neither *u* nor *v* depend on *z*.

***Analysis*** We plug the velocity components into the steady incompressible continuity equation,

*Condition for incompressibility*:

$$\frac{\partial v}{\partial y} = -\underbrace{\frac{\partial u}{\partial x}}_{a} - \underbrace{\frac{\partial w}{\partial z}}_{0} \qquad \frac{\partial v}{\partial y} = -a$$

Next we integrate with respect to *y*. Note that since the integration is a *partial* integration, we must add some arbitrary function of *x* instead of simply a constant of integration.

*Solution*:

$$\boxed{v = -ay + f(x)}$$

If the flow were three-dimensional, we would add a function of *x* and *z* instead.

***Discussion*** To satisfy the incompressible continuity equation, any function of *x* will work since there are no derivatives of *v* with respect to *x* in the continuity equation. Not all functions of *x* are necessarily physically possible, however, since the flow may not be able to satisfy the steady conservation of momentum equation.

---

## 9-33

**Solution** We are to find the *y component* of velocity, *v*, using a given expression for *u*.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane, implying that *w* = 0 and neither *u* nor *v* depend on *z*.

***Analysis*** We plug the velocity components into the steady incompressible continuity equation,

*Condition for incompressibility*:

$$\frac{\partial v}{\partial y} = -\underbrace{\frac{\partial u}{\partial x}}_{2ax-by} - \underbrace{\frac{\partial w}{\partial z}}_{0} \qquad \frac{\partial v}{\partial y} = -2ax + by$$

Next we integrate with respect to *y*. Note that since the integration is a *partial* integration, we must add some arbitrary function of *x* instead of simply a constant of integration.

*Solution*:

$$\boxed{v = -2axy + \frac{by^2}{2} + f(x)}$$

If the flow were three-dimensional, we would add a function of *x* and *z* instead.

***Discussion*** To satisfy the incompressible continuity equation, any function of *x* will work since there are no derivatives of *v* with respect to *x* in the continuity equation.

---

## 9-34

**Solution** For a given axial velocity component in an axisymmetric flow field, we are to generate the radial velocity component.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is axisymmetric implying that $u_\theta = 0$ and there is no variation in the $\theta$ direction.

***Analysis*** We use the incompressible continuity equation in cylindrical coordinates, simplified as follows for axisymmetric flow,

*Incompressible axisymmetric continuity equation*:
$$\frac{1}{r}\frac{\partial(ru_r)}{\partial r}+\frac{\partial(u_z)}{\partial z}=0 \qquad (1)$$

We rearrange Eq. 1,

$$\frac{\partial(ru_r)}{\partial r}=-r\frac{\partial(u_z)}{\partial z}=-r\frac{u_{z,\text{exit}}-u_{z,\text{entrance}}}{L} \qquad (2)$$

We integrate Eq. 2 with respect to *r*,

$$ru_r=-\frac{r^2}{2}\frac{u_{z,\text{exit}}-u_{z,\text{entrance}}}{L}+f(z) \qquad (3)$$

Notice that since we performed a *partial* integration with respect to *r*, we add a function of the other variable *z* rather than simply a constant of integration. We divide all terms in Eq. 3 by *r* and recognize that the term with $f(z)$ will go to infinity at the centerline of the nozzle ($r = 0$) unless $f(z) = 0$. We write our final expression for $u_r$,

*Radial velocity component*:
$$\boxed{u_r=-\frac{r}{2}\frac{u_{z,\text{exit}}-u_{z,\text{entrance}}}{L}} \qquad (4)$$

***Discussion*** You should plug the given equation and Eq. 4 into Eq. 1 to verify that the result is correct. (It is.)

---

## 9-35

**Solution** We are to find the *z component* of velocity using given expressions for *u* and *v*.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible.

***Analysis*** We apply the steady incompressible continuity equation to the given flow field,

*Condition for incompressibility*:
$$\frac{\partial w}{\partial z}=-\underbrace{\frac{\partial u}{\partial x}}_{a+by}-\underbrace{\frac{\partial v}{\partial y}}_{-bz^2} \qquad \frac{\partial w}{\partial z}=-a-by+bz^2$$

Next we integrate with respect to *z*. Note that since the integration is a *partial* integration, we must add some arbitrary function of *x* and *y* instead of simply a constant of integration.

*Solution*:
$$\boxed{w=-az-byz+\frac{bz^3}{3}+f(x,y)}$$

***Discussion*** To satisfy the incompressible continuity equation, any function of *x* and *y* will work since there are no derivatives of *w* with respect to *x* or *y* in the continuity equation.

---

## Stream Function

### 9-36C

**Solution** We are to discuss the significance of curves of constant stream function, and why the stream function is useful.

***Analysis*** Curves of constant stream function represent streamlines of a flow. A stream function is useful because by drawing curves of constant $\psi$, we can visualize the instantaneous velocity field. In addition, the change in the value of $\psi$ from one streamline to another is equal to the volume flow rate per unit width between the two streamlines.

***Discussion*** Streamlines are an instantaneous flow description, as discussed in Chap. 4.

### 9-37C

**Solution** We are to discuss the restrictions on the stream function that cause it to exactly satisfy 2-D incompressible continuity, and why they are necessary.

***Analysis*** **Stream function $\psi$ must be a *smooth function* of *x* and *y* (or *r* and $\theta$)**. These restrictions are necessary so that the second derivatives of $\psi$ with respect to both variables are equal regardless of the order of differentiation. In other words, if $\frac{\partial^2\psi}{\partial x\partial y}=\frac{\partial^2\psi}{\partial y\partial x}$, then the 2-D incompressible continuity equation is satisfied exactly by the definition of $\psi$.

***Discussion*** If the stream function were not smooth, there would be sudden discontinuities in the velocity field as well – a physical impossibility that would violate conservation of mass.

### 9-38C

**Solution** We are to discuss the significance of the difference in value of stream function from one streamline to another.

***Analysis*** **The difference in the value of ψ from one streamline to another is equal to the volume flow rate per unit width between the two streamlines**.

***Discussion*** This fact about the stream function can be used to calculate the volume flow rate in certain applications.

### 9-39

**Solution** For a given stream function we are to generate expressions for the velocity components.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *r*-$\theta$ plane.

***Analysis*** We differentiate $\psi$ to find the velocity components in cylindrical coordinates,

*Radial velocity component*:

$$\boxed{u_\theta=\frac{1}{r}\frac{\partial\psi}{\partial\theta}=V\cos\theta\left(1-\frac{a^2}{r^2}\right)}$$

*Tangential velocity component*:

$$\boxed{u_\theta=-\frac{\partial\psi}{\partial r}=-V\sin\theta\left(1+\frac{a^2}{r^2}\right)}$$

***Discussion*** The radial velocity component is zero at the cylinder surface ($r = a$), but the tangential velocity component is not. In other words, this approximation does not satisfy the no-slip boundary condition along the cylinder surface. See Chap. 10 for a more detailed discussion about such approximations.

**9-40**

**Solution** We are to generate an expression for the stream function along a vertical line in a given flow field, and we are to determine $\psi$ at the top wall.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x-y* plane. **4** The flow is fully developed.

***Analysis*** We start by picking one of the two definitions of the stream function (it doesn't matter which part we choose – the solution will be identical).

$$\frac{\partial \psi}{\partial y} = u = \frac{V}{h} y \qquad (1)$$

Next we integrate Eq. 1 with respect to *y*, noting that this is a *partial* integration and we must add an arbitrary function of the other variable, *x*, rather than a simple constant of integration.

$$\psi = \frac{V}{2h} y^2 + g(x) \qquad (2)$$

Now we choose the other part of the definition of $\psi$, differentiate Eq. 2, and rearrange as follows:

$$v = -\frac{\partial \psi}{\partial x} = -g'(x) \qquad (3)$$

where $g'(x)$ denotes $dg/dx$ since $g$ is a function of only one variable, $x$. We now have two expressions for velocity component $v$, the given equation and Eq. 3. We equate these and integrate with respect to $x$, we find $g(x)$,

$$v = 0 = -g'(x) \qquad g'(x) = 0 \qquad g(x) = C \qquad (4)$$

Note that here we have added an arbitrary constant of integration $C$ since $g$ is a function of $x$ only. Finally, plugging Eq. 4 into Eq. 2 yields the final expression for $\psi$,

*Stream function*:
$$\psi = \frac{V}{2h} y^2 + C \qquad (5)$$

We find constant $C$ by employing the boundary condition on $\psi$. Here, $\psi = 0$ along $y = 0$ (the bottom wall). Thus $C$ is equal to zero by Eq. 5, and

*Stream function*:
$$\boxed{\psi = \frac{V}{2h} y^2} \qquad (6)$$

Along the top wall, $y = h$, and thus

*Stream function along top wall*:
$$\psi_{\text{top}} = \frac{V}{2h} h^2 = \frac{Vh}{2} \qquad (7)$$

***Discussion*** The stream function of Eq. 6 is valid not only along the vertical dashed line of the figure provided in the problem statement, but *everywhere* in the flow since the flow is fully developed and there is nothing special about any particular *x* location.

## 9-41

**Solution** We are to generate an expression for the volume flow rate per unit width for Couette flow. We are to compare results from two methods of calculation.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane. **4** The flow is fully developed.

***Analysis*** We integrate the *x* component of velocity times cross-sectional area to obtain volume flow rate,

$$\dot{V} = \int_A u dA = \int_{y=0}^{y=h} \frac{V}{h} y W dy = \left[ \frac{Vy^2}{2h} W \right]_{y=0}^{y=h} = \frac{Vh}{2} W \qquad (1)$$

where $W$ is the width of the channel into the page. On a per unit width basis, we divide Eq. 1 by $W$ to get

*Volume flow rate per unit width*:
$$\boxed{\frac{\dot{V}}{W} = \frac{Vh}{2}} \qquad (2)$$

The volume flow rate per unit width between any two streamlines $\psi_2$ and $\psi_1$ is equal to $\psi_2 - \psi_1$. We take the streamlines representing the top wall and the bottom wall of the channel. Using the result from the previous problem,

*Volume flow rate per unit width*:
$$\boxed{\frac{\dot{V}}{W} = \psi_{\text{top}} - \psi_{\text{bottom}} = \frac{Vh}{2} - 0 = \frac{Vh}{2}} \qquad (3)$$

Equations 2 and 3 agree, as they must.

***Discussion*** The integration of Eq. 1 can be performed at any *x* location in the channel since the flow is fully developed.

---

## 9-42E

**Solution** We are to plot several streamlines using evenly spaced values of $\psi$ and discuss the spacing between the streamlines.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane. **4** The flow is fully developed.

***Analysis*** The stream function is obtained from the result of Problem 9-40,

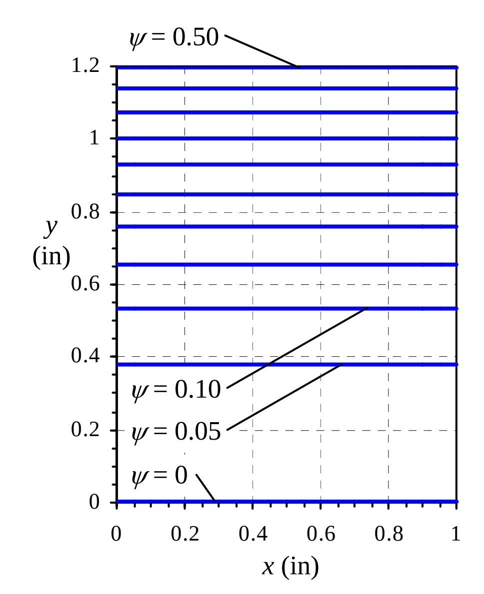
*Stream function*:
$$\psi = \frac{V}{2h} y^2 \qquad (1)$$

We solve Eq. 1 for $y$ as a function of $\psi$ so that we can plot streamlines,

*Equation for streamlines*:
$$y = \sqrt{\frac{2h\psi}{V}} \qquad (2)$$

We have taken only the positive root in Eq. 2 for obvious reasons. Along the top wall, $y = h$, and thus

$$\psi_{\text{top}} = \frac{Vh}{2} = \frac{10.0 \frac{\text{ft}}{\text{s}} \times 0.100 \text{ ft}}{2} = 0.500 \frac{\text{ft}^2}{\text{s}} \qquad (3)$$




The streamlines themselves are straight, flat horizontal lines as seen by Eq. 1. We divide $\psi_{\text{top}}$ by 10 to generate evenly spaced stream functions. We plot 11 streamlines in the figure (counting the streamlines on both walls) by plugging these values of $\psi$ into Eq. 2. **The streamlines are not evenly spaced**. This is because the volume flow rate per unit width between two streamlines $\psi_2$ and $\psi_1$ is equal to $\psi_2 - \psi_1$. The flow speeds near the top of the channel are higher than those near the bottom of the channel, so we expect the streamlines to be closer near the top.

***Discussion*** The extent of the *x* axis in the figure is arbitrary since the flow is fully developed. You can immediately see from a streamline plot like Fig. 1 where flow speeds are high and low (relatively speaking).

---

**9-43**

**Solution** We are to generate an expression for the stream function along a vertical line in a given flow field.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane. **4** The flow is fully developed.

***Analysis*** We start by picking one of the two definitions of the stream function (it doesn't matter which part we choose – the solution will be identical).

$$\frac{\partial \psi}{\partial y} = u = \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right) \quad (1)$$

Next we integrate Eq. 1 with respect to *y*, noting that this is a *partial* integration and we must add an arbitrary function of the other variable, *x*, rather than a simple constant of integration.

$$\psi = \frac{1}{2\mu}\frac{dP}{dx}\left(\frac{y^3}{3} - h\frac{y^2}{2}\right) + g\left(x\right) \quad (2)$$

Now we choose the other part of the definition of $\psi$, differentiate Eq. 2, and rearrange as follows:

$$v = -\frac{\partial \psi}{\partial x} = -g'\left(x\right) \quad (3)$$

where $g'(x)$ denotes $dg/dx$ since *g* is a function of only one variable, *x*. We now have two expressions for velocity component *v*, the given equation and Eq. 3. We equate these and integrate with respect to *x* to find $g(x)$,

$$v = 0 = -g'\left(x\right) \qquad g'\left(x\right) = 0 \qquad g\left(x\right) = C \quad (4)$$

Note that here we have added an arbitrary constant of integration *C* since *g* is a function of *x* only. Finally, plugging Eq. 4 into Eq. 2 yields the final expression for $\psi$,

*Stream function*:

$$\psi = \frac{1}{2\mu}\frac{dP}{dx}\left(\frac{y^3}{3} - h\frac{y^2}{2}\right) + C \quad (5)$$

We find constant *C* by employing the boundary condition on $\psi$. Here, $\psi = 0$ along $y = 0$ (the bottom wall). Thus *C* is equal to zero by Eq. 5, and

*Stream function*:

$$\boxed{\psi = \frac{1}{2\mu}\frac{dP}{dx}\left(\frac{y^3}{3} - h\frac{y^2}{2}\right)} \quad (6)$$

Along the top wall, $y = h$, and thus

*Stream function along top wall*:

$$\boxed{\psi_{\text{top}} = -\frac{1}{12\mu}\frac{dP}{dx}h^3} \quad (7)$$

***Discussion*** The stream function of Eq. 6 is valid not only along the vertical dashed line of the figure provided in the problem statement, but *everywhere* in the flow since the flow is fully developed and there is nothing special about any particular *x* location.

**9-44**

**Solution** We are to generate an expression for the volume flow rate per unit width for fully developed channel flow. We are to compare results from two methods of calculation.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane. **4** The flow is fully developed.

***Analysis*** We integrate the *x* component of velocity times cross-sectional area to obtain volume flow rate,

$$\dot{V} = \int_A u dA = \int_{y=0}^{y=h} \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right)W dy = \left[\frac{1}{2\mu}\frac{dP}{dx}\left(\frac{y^3}{3} - h\frac{y^2}{2}\right)W\right]_{y=0}^{y=h}$$
$$= \frac{1}{2\mu}\frac{dP}{dx}\left(-\frac{h^3}{6}\right)W = -\frac{1}{12\mu}\frac{dP}{dx}h^3 W \qquad (1)$$

where *W* is the width of the channel into the page. On a per unit width basis, we divide Eq. 1 by *W* to get

*Volume flow rate per unit width*:
$$\boxed{\frac{\dot{V}}{W} = -\frac{1}{12\mu}\frac{dP}{dx}h^3} \qquad (2)$$

The volume flow rate per unit width between any two streamlines $\psi_2$ and $\psi_1$ is equal to $\psi_2 - \psi_1$. We take the streamlines representing the top wall and the bottom wall of the channel. Using the result from the previous problem,

*Volume flow rate per unit width*:
$$\boxed{\frac{\dot{V}}{W} = \psi_{\text{top}} - \psi_{\text{bottom}} = -\frac{1}{12\mu}\frac{dP}{dx}h^3 - 0 = -\frac{1}{12\mu}\frac{dP}{dx}h^3} \qquad (3)$$

Equations 2 and 3 agree, as they must.

***Discussion*** The integration of Eq. 1 can be performed at any *x* location in the channel since the flow is fully developed.

**9-45**

**Solution** We are to plot several streamlines using evenly spaced values of $\psi$ and discuss the spacing between the streamlines.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane. **4** The flow is fully developed.

***Properties*** The viscosity of water at $T$ = 20$^o$C is 1.002 × 10$^{-3}$ kg/(m·s).

***Analysis*** The stream function is obtained from the result of Problem 9-43,

*Stream function*:
$$\psi = \frac{1}{2\mu}\frac{dP}{dx}\left(\frac{y^3}{3} - h\frac{y^2}{2}\right) \qquad (1)$$

We need to solve Eq. 1 (a cubic equation) for *y* as a function of $\psi$ so that we can plot streamlines. First we re-write Eq. 1 in standard cubic form,

*Standard cubic form*:
$$y^3 - \frac{3h}{2}y^2 - \frac{6\mu\psi}{dP/dx} = 0 \qquad (2)$$

We can either look up the solution for cubic equations or use Newton's iteration method to obtain *y* for a given value of $\psi$. In general there are three roots – we choose the positive real root with $0 < y < h$, which is the only one that has physical meaning for this problem. Along the top wall, $y = h$, and Eq. 1 yields

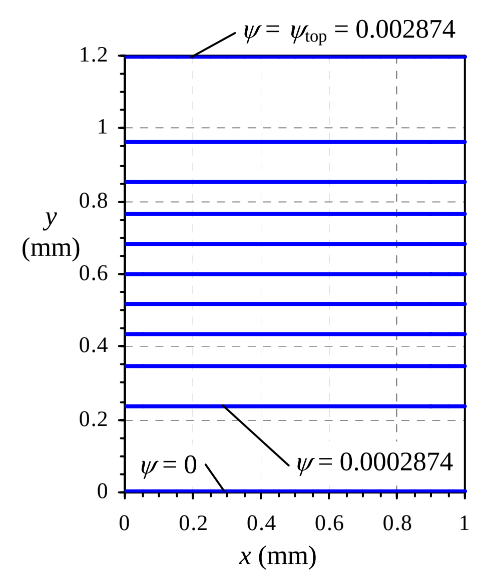



**FIGURE 1**
Streamlines for 2-D channel flow with evenly spaced values of stream function. Values of $\psi$ are in units of m$^2$/s.

*Stream function along top wall*:
$$\psi_{top} = -\frac{1}{12\mu}\frac{dP}{dx}h^3 = \frac{1}{12\left(1.002\times10^{-3}\ \text{kg/m}\cdot\text{s}\right)}\left(20,000\ \text{N/m}^3\right)\left(0.00120\ \text{m}\right)^3\left(\frac{\text{kg}\cdot\text{m}}{\text{s}^2\text{N}}\right)$$
$$= 2.874\times10^{-3}\ \text{m}^2/\text{s}$$

The streamlines themselves are straight, flat horizontal lines as seen by Eq. 1. We divide $\psi_{top}$ by 10 to generate evenly spaced stream functions. We plot 11 streamlines in Fig. 1 (counting the streamlines on both walls) by plugging these values of $\psi$ into Eq. 2 and solving for *y*.

**The streamlines are not evenly spaced**. This is because the volume flow rate per unit width between two streamlines $\psi_2$ and $\psi_1$ is equal to $\psi_2 - \psi_1$. The flow speeds in the middle of the channel are higher than those near the top or bottom of the channel, so we expect the streamlines to be closer near the middle.

***Discussion*** The extent of the *x* axis in Fig. 1 is arbitrary since the flow is fully developed. You can immediately see from a streamline plot like Fig. 1 where flow speeds are high and low (relatively speaking).

## 9-46

**Solution** We are to calculate the volume flow rate and average speed of air being sucked through a sampling probe.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional.

***Analysis*** For 2-D incompressible flow the difference in the value of the stream function between two streamlines is equal to the volume flow rate per unit width between the two streamlines. Thus,

*Volume flow rate through the sampling probe*:
$$\dot{V} = (\psi_u - \psi_l)\times W = (0.150 - 0.105)\frac{\text{m}^2}{\text{s}}\times 0.052\text{ m} = \mathbf{0.00234\ m^3/s} \quad (1)$$

The average speed of air in the probe is obtained by dividing volume flow rate by cross-sectional area,

*Average speed through the sampling probe*:
$$V_{\text{avg}} = \frac{\dot{V}}{hW} = \frac{0.00234\text{ m}^3/\text{s}}{(0.0045\text{ m})(0.052\text{ m})} = \mathbf{10.0\ m/s} \quad (2)$$

***Discussion*** Notice that the streamlines inside the probe are more closely packed than are those outside the probe because the flow speed is higher inside the probe.

## 9-47

**Solution** We are to sketch streamlines for the case of a sampling probe with too little suction, and we are to name this type of sampling and label the lower and upper dividing streamlines.

***Analysis*** If the suction were too weak, the volume flow rate through the probe would be too low and the average air speed through the probe would be lower than that of the air stream. The dividing streamlines would diverge outward rather than inward as sketched in Fig. 1. We would call this type of sampling **subisokinetic sampling**.

***Discussion*** We have drawn the streamlines inside the probe further apart than those in the air stream because the flow speed is lower inside the probe.

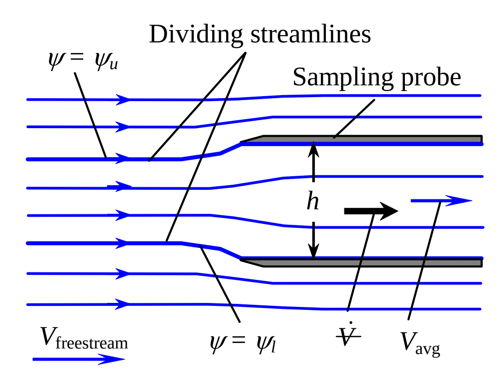



**FIGURE 1**
Streamlines for *subisokinetic sampling*.

## 9-48

**Solution** We are to calculate the speed of the air stream of Fig. P9-46.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional.

***Analysis*** In the air stream far upstream of the probe,

*Volume flow rate per unit width*:
$$\frac{\dot{V}}{W} = \psi_u - \psi_l = V_{\text{freestream}}y_u - V_\infty y_l = V_{\text{freestream}}(y_u - y_l) \quad (1)$$

By definition of streamlines, the volume flow rate between the two dividing streamlines must be the same as that through the probe itself. We know the volume flow rate through the probe from the results of Problem 9-46. The value of the stream function on the lower and upper dividing streamlines are the same as those of Problem 9-46, namely $\psi_l$ = 0.105 m$^2$/s and $\psi_u$ = 0.150 m$^2$/s respectively. We also know $y_u - y_l$ from the information given here. Thus, Eq. 1 yields

*Freestream speed*:
$$V_{\text{freestream}} = \frac{\psi_u - \psi_l}{(y_u - y_l)} = \frac{(0.150 - 0.105)\frac{\text{m}^2}{\text{s}}}{0.0058\text{ m}} = \mathbf{7.76\ \frac{m}{s}} \quad (2)$$

***Discussion*** We verify by these calculations that the sampling is superisokinetic (average speed through the probe is higher than that of the upstream air stream).

**9-49**

**Solution** For a given velocity field we are to generate an expression for $\psi$, and we are to calculate the volume flow rate per unit width between two streamlines.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x-y* plane.

***Analysis*** We start by picking one of the two definitions of the stream function (it doesn't matter which part we choose – the solution will be identical).

$$\frac{\partial \psi}{\partial y} = u = V \qquad (1)$$

Next we integrate Eq. 1 with respect to $y$, noting that this is a *partial* integration and we must add an arbitrary function of the other variable, $x$, rather than a simple constant of integration.

$$\psi = Vy + g(x) \qquad (2)$$

Now we choose the other part of the definition of $\psi$, differentiate Eq. 2, and rearrange as follows:

$$v = -\frac{\partial \psi}{\partial x} = -g'(x) \qquad (3)$$

where $g'(x)$ denotes $dg/dx$ since $g$ is a function of only one variable, $x$. We now have two expressions for velocity component $v$, the given equation and Eq. 3. We equate these and integrate with respect to $x$ to find $g(x)$,

$$v = 0 = -g'(x) \qquad g'(x) = 0 \qquad g(x) = C \qquad (4)$$

Note that here we have added an arbitrary constant of integration $C$ since $g$ is a function of $x$ only. Finally, plugging Eq. 4 into Eq. 2 yields the final expression for $\psi$,

*Stream function*: $$\boxed{\psi = Vy + C} \qquad (5)$$

Constant $C$ is arbitrary; it is common to set it to zero, although it can be set to any desired value. Here, $\psi = 0$ along the streamline at $y = 0$, forcing $C$ to equal zero by Eq. 5. For the streamline at $y = 0.5$ m,

*Value of* $\psi_2$: $$\psi_2 = \left(8.9\ \frac{\text{m}}{\text{s}}\right) \times (0.5\ \text{m}) = 4.45\frac{\text{m}^2}{\text{s}} \qquad (6)$$

The volume flow rate per unit width between streamlines $\psi_2$ and $\psi_0$ is equal to $\psi_2 - \psi_0$,

*Volume flow rate per unit width*: $$\frac{\dot{V}}{W} = \psi_2 - \psi_0 = (4.45 - 0)\frac{\text{m}^2}{\text{s}} = \mathbf{4.45\frac{m^2}{s}} \qquad (7)$$

We verify our result by calculating the volume flow rate per unit width from first principles. Namely, volume flow rate is equal to speed times cross-sectional area,

*Volume flow rate per unit width*:

$$\frac{\dot{V}}{W} = V(y_2 - y_0) = 8.9\frac{\text{m}}{\text{s}} \times (0.5 - 0)\ \text{m} = \mathbf{4.45\frac{m^2}{s}} \qquad (8)$$

***Discussion*** If constant $C$ were some value besides zero, we would still get the same result for the volume flow rate since $C$ would cancel out in the subtraction.

**9-50E**

**Solution** For a given velocity field we are to generate an expression for $\psi$ and plot several streamlines for given values of constants $a$ and $b$.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the $x$-$y$ plane, implying that $w = 0$ and neither $u$ nor $v$ depend on $z$.

***Analysis*** We plug the given equation into the steady incompressible continuity equation,

*Condition for incompressibility*: $$\frac{\partial v}{\partial y} = -\underbrace{\frac{\partial u}{\partial x}}_{2ax-by} - \underbrace{\frac{\partial w}{\partial z}}_{0} \qquad \frac{\partial v}{\partial y} = -2ax + by$$

Next we integrate with respect to $y$. Note that since the integration is a *partial* integration, we must add some arbitrary function of $x$ instead of simply a constant of integration.

*y component of velocity*: $$v = -2axy + \frac{by^2}{2} + f(x)$$

If the flow were three-dimensional, we would add a function of $x$ and $z$ instead. We are told that $v = 0$ for all values of $x$ when $y = 0$. This is only possible if $f(x) = 0$. Thus,

*y component of velocity*: $$v = -2axy + \frac{by^2}{2} \qquad (1)$$

To obtain the stream function, we start by picking one of the two parts of the definition of the stream function,

$$\frac{\partial \psi}{\partial y} = u = ax^2 - bxy$$

Next we integrate the above equation with respect to $y$, noting that this is a *partial* integration and we must add an arbitrary function of the other variable, $x$, rather than a simple constant of integration.

$$\psi = ax^2 y - \frac{bxy^2}{2} + g(x) \qquad (2)$$

Now we choose the other part of the definition of $\psi$, differentiate Eq. 2, and rearrange as follows:

$$v = -\frac{\partial \psi}{\partial x} = -2axy + \frac{by^2}{2} - g'(x) \qquad (3)$$

where $g'(x)$ denotes $dg/dx$ since $g$ is a function of only one variable, $x$. We now have two expressions for velocity component $v$, Eq. 1 and Eq. 3. We equate these and integrate with respect to $x$ to find $g(x)$,

$$g'(x) = 0 \qquad g(x) = C \qquad (4)$$

Note that here we have added an arbitrary constant of integration $C$ since $g$ is a function of $x$ only. But C must be zero in order for $\psi$ to be zero for any value of $x$ when $y = 0$. Finally, Eq. 2 yields the final expression for $\psi$,

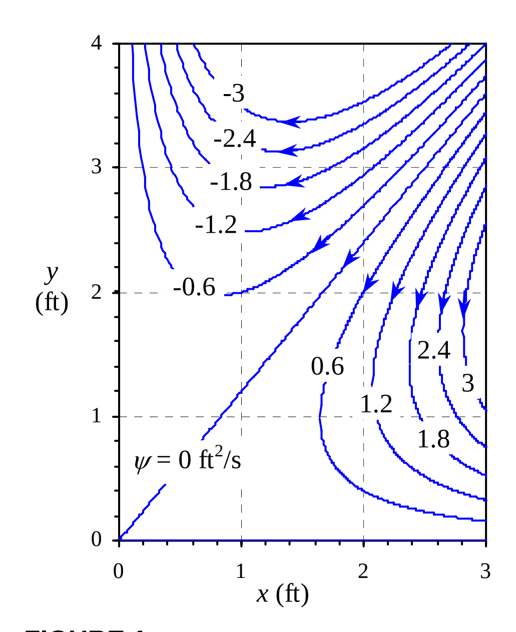



**FIGURE 1**
Streamlines for a given velocity field; the value of constant $\psi$ is indicated for each streamline in units of $ft^2$/s.

*Solution*: $$\boxed{\psi = ax^2 y - \frac{bxy^2}{2}} \qquad (5)$$

To plot the streamlines, we note that Eq. 5 represents a *family* of curves, one unique curve for each value of the stream function $\psi$. We solve Eq. 5 for $y$ as a function of $x$. A bit of algebra (the quadratic rule) yields

*Equation for streamlines*:
$$y = \frac{ax^2 \pm \sqrt{a^2x^4 - 2\psi bx}}{bx} \qquad (6)$$

For the given values of constants $a$ and $b$, we plot Eq. 6 for several values of $\psi$ in Fig. 1; these curves of constant $\psi$ are streamlines of the flow. Note that both the positive and negative roots of Eq. 6 are required to plot each streamline. The direction of the flow is found by calculating $u$ and $v$ at some point in the flow field. We pick $x$ = 2 ft, $y$ = 2 ft, where $u$ = –1.2 ft/s and $v$ = –2.1 ft/s. This indicates flow to the lower left near this location. We fill in the rest of the arrows in Fig. 1 to be consistent. We see that the flow enters from the upper right, and splits into two parts – one to the lower right and one to the upper left.

***Discussion*** It is always a good idea to check your algebra. In this example, you should differentiate Eq. 5 to verify that the velocity components of the given equation are obtained.

---

## 9-51

**Solution** For a given velocity field we are to generate an expression for $\psi$.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the $x$-$y$ plane.

***Analysis*** We start by picking one of the two definitions of the stream function (it doesn't matter which part we choose – the solution will be identical).

$$\frac{\partial \psi}{\partial y} = u = V\cos\alpha \qquad (1)$$

Next we integrate Eq. 1 with respect to $y$, noting that this is a *partial* integration and we must add an arbitrary function of the other variable, $x$, rather than a simple constant of integration.

$$\psi = yV\cos\alpha + g(x) \qquad (2)$$

Now we choose the other part of the definition of $\psi$, differentiate Eq. 2, and rearrange as follows:

$$v = -\frac{\partial \psi}{\partial x} = -g'(x) \qquad (3)$$

where $g'(x)$ denotes $dg/dx$ since $g$ is a function of only one variable, $x$. We now have two expressions for velocity component $v$, the given equation and Eq. 3. We equate these and integrate with respect to $x$ to find $g(x)$,

$$v = V\sin\alpha = -g'(x) \qquad g'(x) = -V\sin\alpha \qquad g(x) = -xV\sin\alpha + C \qquad (4)$$

Note that here we have added an arbitrary constant of integration $C$ since $g$ is a function of $x$ only. Finally, plugging Eq. 4 into Eq. 2 yields the final expression for $\psi$,

*Stream function*:
$$\boxed{\psi = V\left(y\cos\alpha - x\sin\alpha\right) + C} \qquad (5)$$

Constant $C$ is arbitrary; it is common to set it to zero, although it can be set to any desired value.

***Discussion*** You can verify by differentiating $\psi$ that Eq. 5 yields the correct values of $u$ and $v$.

---

## 9-52

**Solution** For a given stream function, we are to calculate the velocity components and verify incompressibility.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible (this assumption is to be verified). **3** The flow is two-dimensional in the *x*-*y* plane, implying that $w = 0$ and neither *u* nor *v* depend on *z*.

***Analysis*** (*a*) We use the definition of $\psi$ to obtain expressions for *u* and *v*.

*Velocity components*:
$$\boxed{u = \frac{\partial \psi}{\partial y} = bx + 2cy \qquad v = -\frac{\partial \psi}{\partial x} = -2ax - by} \qquad (1)$$

(*b*) We check if the incompressible continuity equation in the *x*-*y* plane is satisfied by the velocity components of Eq. 1,

*Incompressible continuity*:
$$\underbrace{\frac{\partial u}{\partial x}}_{b} + \underbrace{\frac{\partial v}{\partial y}}_{-b} + \underbrace{\frac{\partial w}{\partial z}}_{0} = 0 \qquad b - b = 0 \qquad (2)$$

*We conclude that the flow is indeed incompressible.*

***Discussion*** Since $\psi$ is a smooth function of *x* and *y*, it automatically satisfies the continuity equation by its definition. Equation 2 confirms this. If it did not, we would go back and look for an algebra mistake somewhere.

---

## 9-53 [*Also solved using EES on enclosed DVD*]

**Solution** We are to plot several streamlines for a given velocity field.

***Analysis*** We re-write the stream function equation of the previous problem with all the terms on one side,

$$cy^2 + bxy + ax^2 - \psi = 0 \qquad (1)$$

For any constant value of $\psi$ (along a streamline), Eq. 1 is in a form that enables us to use the quadratic rule to solve for *y* as a function of *x*,

*Equation for a streamline*:
$$y = \frac{-bx \pm \sqrt{b^2x^2 - 4c\left(ax^2 - \psi\right)}}{2c} \qquad (2)$$

We plot the streamlines in Fig. 1. For each value of $\psi$ there are two curves – one for the positive root and one for the negative root of Eq. 2. There is symmetry about a diagonal line through the origin. The streamlines appear to be hyperbolae. We determine the flow direction by plugging in a couple values of *x* and *y* and calculating the velocity components; e.g., at *x* = 1 m and *y* = 3 m, *u* = 2.7 m/s and *v* = 4.9 m/s. The flow at this point is in the upper right direction. Similarly, at *x* = 1 m and *y* = -2 m, *u* = -3.3 m/s and *v* = -1.6 m/s. The flow at this point is in the lower left direction.

***Discussion*** This flow may not represent any particular physical flow field, but it produces an interesting flow pattern.

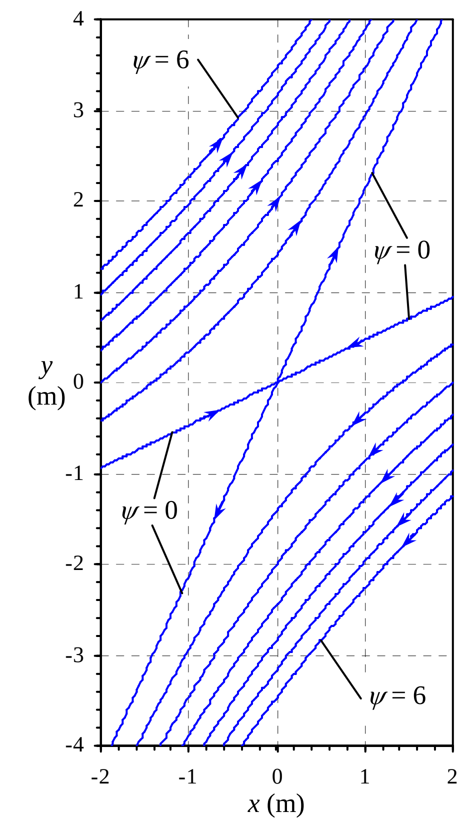



**FIGURE 1**
Streamlines for a given velocity field. Values of $\psi$ are in units of $m^2/s$.

---

## 9-54

**Solution** For a given stream function, we are to calculate the velocity components and verify incompressibility.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane, implying that $w = 0$ and neither *u* nor *v* depend on *z*.

***Analysis*** (*a*) We use the definition of $\psi$ to obtain expressions for *u* and *v*.

*Velocity components*:
$$\boxed{u = \frac{\partial \psi}{\partial y} = -2by + dx \qquad v = -\frac{\partial \psi}{\partial x} = -2ax - c - dy} \qquad (1)$$

(*b*) We check if the incompressible continuity equation in the *x*-*y* plane is satisfied by the velocity components of Eq. 1,

*Incompressible continuity*:
$$\underbrace{\frac{\partial u}{\partial x}}_{d} + \underbrace{\frac{\partial v}{\partial y}}_{-d} + \underbrace{\frac{\partial w}{\partial z}}_{0} = 0 \qquad d - d = 0 \qquad (2)$$

**We conclude that the flow is indeed incompressible**.

***Discussion*** Since $\psi$ is a smooth function of *x* and *y*, it automatically satisfies the continuity equation by its definition. Eq. 2 confirms this. If it did not, we would go back and look for an algebra mistake somewhere.

## 9-55

**Solution** We are to make up a stream function $\psi(x,y)$, calculate the velocity components and verify incompressibility.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** Every student should have a different stream function. He or she then takes the derivatives with respect to *y* and *x* to find *u* and *v*. The student should then plug his/her velocity components into the incompressible continuity equation. **Continuity will be satisfied regardless of $\psi(x,y)$, provided that $\psi(x,y)$ is a smooth function of *x* and *y*.**

***Discussion*** As long as $\psi$ is a smooth function of *x* and *y*, it automatically satisfies the continuity equation by its definition.

## 9-56

**Solution** We are to calculate the percentage of flow going through one branch of a branching duct.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** For 2-D incompressible flow the difference in the value of the stream function between two streamlines is equal to the volume flow rate per unit width between the two streamlines. Thus,

*Main branch*:
$$\left.\frac{\dot{V}}{W}\right)_{\text{main}} = \psi_{\text{upper wall}} - \psi_{\text{lower wall}} = (4.15 - 2.03)\frac{\text{m}^2}{\text{s}} = 2.12\frac{\text{m}^2}{\text{s}} \qquad (1)$$

Similary, in the upper branch,

*Upper branch*:
$$\left.\frac{\dot{V}}{W}\right)_{\text{upper}} = \psi_{\text{upper wall}} - \psi_{\text{branch wall}} = (4.15 - 2.80)\frac{\text{m}^2}{\text{s}} = 1.35\frac{\text{m}^2}{\text{s}} \qquad (2)$$

On a percentage basis, the percentage of volume flow through the upper branch is calculated as

$$\frac{\dot{V}_{\text{upper}}}{\dot{V}_{\text{main}}} = \frac{\left.\frac{\dot{V}}{W}\right)_{\text{upper}}}{\left.\frac{\dot{V}}{W}\right)_{\text{main}}} = \frac{1.35\frac{\text{m}^2}{\text{s}}}{2.12\frac{\text{m}^2}{\text{s}}} = 0.637 = \mathbf{63.7\%} \qquad (3)$$

***Discussion*** No dimensions are given, so it is not possible to calculate velocities.

**9-57**

**Solution** We are to calculate duct height *h* for a given average velocity through a duct and values of stream function along the duct walls.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** The volume flow rate through the main branch of the duct is equal to the average velocity times the cross-sectional area of the duct,

*Volume flow rate*:
$$\dot{V} = V_{\text{avg}} W h \qquad (1)$$

We solve for h in Eq. 1, using the results of Problem 9-56,

*Duct height*:
$$h = \frac{1}{V_{\text{avg}}}\frac{\dot{V}}{W}\bigg)_{\text{main}} = \frac{1}{11.4\ \frac{\text{m}}{\text{s}}} \times 2.12\ \frac{\text{m}^2}{\text{s}}\left(\frac{100\ \text{cm}}{\text{m}}\right) = \mathbf{18.6\ cm} \qquad (2)$$

An alternative way to solve for height *h* is to assume uniform flow in the main branch, for which $\psi = V_{\text{avg}}y$. We take the difference between $\psi$ at the top of the duct and $\psi$ at the bottom of the duct to find *h*,

$$\psi_{\text{upper wall}} - \psi_{\text{lower wall}} = V_{\text{avg}} y_{\text{upper wall}} - V_{\text{avg}} y_{\text{lower wall}} = V_{\text{avg}}\left(y_{\text{upper wall}} - y_{\text{lower wall}}\right) = V_{\text{avg}} h$$

Thus,

*Duct height*:
$$h = \frac{\psi_{\text{upper wall}} - \psi_{\text{lower wall}}}{V_{\text{avg}}} = \frac{(4.15 - 2.03)\ \frac{\text{m}^2}{\text{s}}}{11.4\ \frac{\text{m}}{\text{s}}}\left(\frac{100\ \text{cm}}{\text{m}}\right) = \mathbf{18.6\ cm} \qquad (3)$$

You can see that we get the same result as that of Eq. 2.

***Discussion*** The result is correct even if the velocity profile through the duct is not uniform, since we have used the *average* velocity in our calculations.

---

**9-58**

**Solution** We are to verify that the given $\psi$ satisfies the continuity equation, and we are to discuss any restrictions.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is axisymmetric ($\psi$ is a function of *r* and *z* only).

***Analysis*** We plug the given velocity components into the axisymmetric continuity equation,

$$\frac{1}{r}\frac{\partial(ru_r)}{\partial r} + \frac{\partial(u_z)}{\partial z} = \frac{1}{r}\frac{\partial\left(-\frac{\partial\psi}{\partial z}\right)}{\partial r} + \frac{\partial\left(\frac{1}{r}\frac{\partial\psi}{\partial r}\right)}{\partial z} = \frac{1}{r}\left(-\frac{\partial^2\psi}{\partial r\partial z} + \frac{\partial^2\psi}{\partial z\partial r}\right) = 0$$

Thus we see that **continuity is satisfied by the given stream function**. The only restriction on $\psi$ is that $\boldsymbol{\psi}$ **must be a smooth function of *r* and *z***.

***Discussion*** For a smooth function of two variables, the order of differentiation does not matter.

---

## 9-59

**Solution** We are to determine the value of the stream function along the positive *y* axis and the negative *x* axis for the case of a line source at the origin.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* or *r*-$\theta$ plane.

***Analysis*** For 2-D incompressible flow the difference in the value of the stream function between two streamlines is equal to the volume flow rate per unit width between the two streamlines. Let us take the arc of the circle of radius *r* between the positive *x* axis and the positive *y* axis of the figure provided with the problem statement. The volume flow rate per unit width through this arc is one-fourth of $\dot{V}/L$, the total volume flow rate per unit width, since the arc spans exactly one-fourth of the circumference of the circle.

$$\psi_{\text{positive } y \text{ axis}} - \psi_{\text{positive } x \text{ axis}} = \frac{1}{4}\frac{\dot{V}}{L} \qquad (1)$$

Since $\psi = 0$ along the positive *x* axis, we conclude that

*$\psi$ along positive y axis*:

$$\boxed{\psi_{\text{positive } y \text{ axis}} = \frac{1}{4}\frac{\dot{V}}{L}} \qquad (2)$$

Similarly, the volume flow rate through the top half of the circle is half of the total volume flow rate and we conclude that

*$\psi$ along negative x axis*:

$$\boxed{\psi_{\text{negative } y \text{ axis}} = \frac{1}{2}\frac{\dot{V}}{L}} \qquad (3)$$

***Discussion*** Some CFD codes use $\psi$ as a variable, and we thus need to specify the value of $\psi$ along boundaries of the computational domain. Simple calculations such as this are useful in these situations.

---

## 9-60

**Solution** We are to determine the value of the stream function along the positive *y* axis and the negative *x* axis for the case of a line sink at the origin.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* or *r*-$\theta$ plane.

***Analysis*** Everything is the same as in Problem 9-59 except that the flow direction is reversed everywhere. The volume flow rate per unit width through the arc of radius *r* between the positive *x* axis and the positive *y* axis of Fig. P9-59 is now *negative* one-fourth of $\dot{V}/L$ since the flow is now mathematically negative.

$$\psi_{\text{positive } y \text{ axis}} - \psi_{\text{positive } x \text{ axis}} = -\frac{1}{4}\frac{\dot{V}}{L} \qquad (1)$$

Since $\psi = 0$ along the positive *x* axis, we conclude that

*$\psi$ along positive y axis*:

$$\boxed{\psi_{\text{positive } y \text{ axis}} = -\frac{1}{4}\frac{\dot{V}}{L}} \qquad (2)$$

Similarly, the volume flow rate through the top half of the circle is half of the total volume flow rate and is negative. We conclude that

*$\psi$ along negative x axis*:

$$\boxed{\psi_{\text{negative } x \text{ axis}} = -\frac{1}{2}\frac{\dot{V}}{L}} \qquad (3)$$

***Discussion*** We need to be careful of the sign of $\psi$.

---

**9-61**

**Solution** We are to generate an expression for the stream function that describes a given velocity field.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is axisymmetric ($\psi$ is a function of $r$ and $z$ only).

***Analysis*** The $r$ and $z$ velocity components from Problem 9-34 are

*Velocity field*:
$$u_r = -\frac{r}{2}\frac{u_{z,\text{exit}} - u_{z,\text{entrance}}}{L} \qquad u_z = u_{z,\text{entrance}} + \frac{u_{z,\text{exit}} - u_{z,\text{entrance}}}{L}z \qquad (1)$$

To generate the stream function we use the definition of $\psi$ for steady, incompressible, axisymmetric flow,

*Axisymmetric stream function*:
$$u_r = -\frac{1}{r}\frac{\partial\psi}{\partial z} \qquad u_z = \frac{1}{r}\frac{\partial\psi}{\partial r} \qquad (2)$$

We choose one of the definitions of Eq. 2 to integrate. We pick the second one,

*Integration*:
$$\begin{aligned}\psi &= \int r u_z dr = \int r\left(u_{z,\text{entrance}} + \frac{u_{z,\text{exit}} - u_{z,\text{entrance}}}{L}z\right)dr \\ &= \frac{r^2}{2}\left(u_{z,\text{entrance}} + \frac{u_{z,\text{exit}} - u_{z,\text{entrance}}}{L}z\right) + f(z)\end{aligned} \qquad (3)$$

We added a function of $z$ instead of a constant of integration since this is a partial integration. Now we take the $z$ derivative of Eq. 3 and use the other half of Eq. 2,

*Differentiation*:
$$u_r = -\frac{1}{r}\frac{\partial\psi}{\partial z} = -\frac{r}{2}\frac{u_{z,\text{exit}} - u_{z,\text{entrance}}}{L} - \frac{1}{r}f'(z) \qquad (4)$$

We equate Eq. 4 to the known value of $u_r$ from Eq. 1,

*Comparison*:
$$u_r = -\frac{r}{2}\frac{u_{z,\text{exit}} - u_{z,\text{entrance}}}{L} - \frac{1}{r}f'(z) = -\frac{r}{2}\frac{u_{z,\text{exit}} - u_{z,\text{entrance}}}{L} \quad \text{or} \quad f'(z) = 0 \qquad (5)$$

Since $f$ is a function of $z$ only, integration of Eq. 5 yields $f(z)$ = constant. The final result is thus

*Stream function*:
$$\boxed{\psi = \frac{r^2}{2}\left(u_{z,\text{entrance}} + \frac{u_{z,\text{exit}} - u_{z,\text{entrance}}}{L}z\right) + \text{constant}} \qquad (6)$$

***Discussion*** The constant of integration can be any value since velocity components are determined by taking *derivatives* of the stream function.

**9-62E** [*Also solved using EES on enclosed DVD*]

**Solution** We are to calculate the axial speed at the entrance and exit of the nozzle, and we are to plot several streamlines for a given axisymmetric flow field.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is axisymmetric ($\psi$ is a function of $r$ and $z$ only).

***Analysis*** (*a*) Since $u_z$ is not a function of radius, the axial velocity profile across a cross section of the nozzle is uniform. (This is consistent with the assumption that frictional effects along the nozzle walls are neglected.) Thus, at any cross section the axial speed is equal to the volume flow rate divided by cross-sectional area,

*Entrance axial speed*:

$$u_{z,\text{entrance}} = \frac{4\dot{V}}{\pi D_{\text{entrance}}^2} = \frac{4\times 2.0\,\frac{\text{gal}}{\text{min}}}{\pi\left(0.50\text{ in}\right)^2}\left(\frac{0.1337\text{ ft}^3}{\text{gal}}\right)\left(\frac{12\text{ in}}{\text{ft}}\right)^2\left(\frac{\text{min}}{60\text{ s}}\right) = \mathbf{3.268\,\frac{ft}{s}} \quad (1)$$

Similarly,

*Exit axial speed*:

$$u_{z,\text{exit}} = \frac{4\dot{V}}{\pi D_{\text{exit}}^2} = \mathbf{41.69\,\frac{ft}{s}} \quad (2)$$

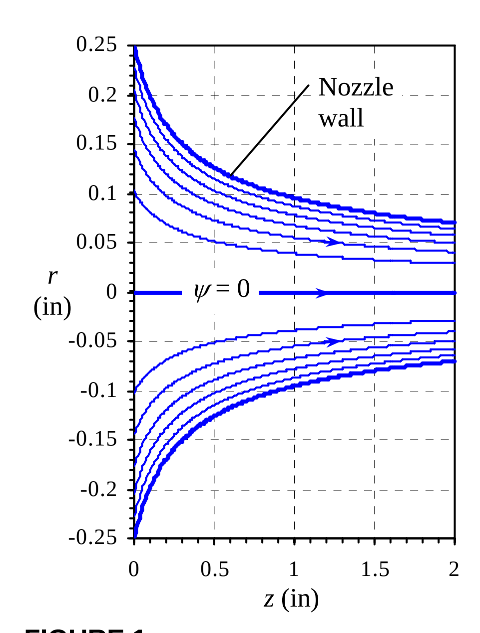



**FIGURE 1**
Streamlines for flow through an axisymmetric garden hose nozzle. Note that the vertical axis is highly magnified.

(*b*) We use the stream function developed in Problem 9-61. Setting the constant to zero for simplicity, we have

*Stream function*:

$$\psi = \frac{r^2}{2}\left(u_{z,\text{entrance}} + \frac{u_{z,\text{exit}} - u_{z,\text{entrance}}}{L}z\right) \quad (3)$$

We solve Eq. 3 for $r$ as a function of $z$ and plot several streamlines in Fig. 1,

*Streamlines*:

$$r = \pm\sqrt{\frac{2\psi}{u_{z,\text{entrance}} + \frac{u_{z,\text{exit}} - u_{z,\text{entrance}}}{L}z}} \quad (4)$$

At the nozzle entrance ($z$ = 0), the wall is at $r = D_{\text{entrance}}/2$ = 0.25 inches. Eq. 3 yields $\psi_{\text{wall}}$ = 0.0007073 ft$^3$/s for the streamline that passes through this point. This streamline thus represents the shape of the nozzle wall, and we have designed the nozzle shape.

***Discussion*** You can verify that the diameter between the outermost streamlines varies from $D_{\text{entrance}}$ to $D_{\text{exit}}$.

## 9-63

**Solution** We are to discuss the sign of the stream function in a separation bubble, and determine where $\psi$ is a minimum.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** For 2-D incompressible flow the difference in the value of the stream function between two streamlines is equal to the volume flow rate per unit width between the two streamlines. For example, $\psi_{upper} - \psi_0$ is positive and represents the volume flow rate per unit width between the wall and the uppermost streamline, The flow between these two streamlines is to the right. Likewise, the difference between $\psi$ along the dividing streamline and $\psi = \psi_1$ along a streamline in the upper part of the separation bubble must also be positive as sketched in Fig. 1. The arc-shaped dividing streamline divides fluid within the separation bubble from fluid outside of the separation bubble. The stream function along this dividing streamline must be zero since it intersects the wall where $\psi = 0$. The only way we can have flow to the right in the upper part of the separation bubble is if $\psi_1$ is *negative* (Fig. 1). We conclude that **for this problem, all streamlines within the separation bubble have negative values of stream function**. **The minimum value of $\psi$ occurs in the center of the separation bubble as sketched in Fig. 1**.

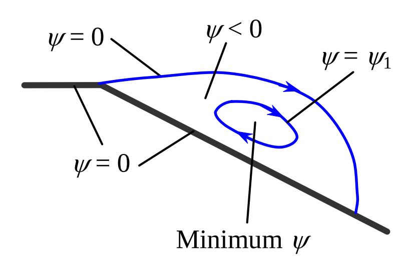



**FIGURE 1**
Close-up of streamlines near the separation bubble. The minimum value of the stream function occurs in the middle of the separation bubble.

***Discussion*** We cannot conclude that $\psi$ is always negative within a separation bubble, since we can add any arbitrary constant to all the $\psi$ values, and it will not change the flow.

## 9-64

**Solution** We are to discuss how someone can interpret the relative speed of a flow based solely on contours of constant stream function.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible.

***Analysis*** For 2-D incompressible flow the difference in the value of the stream function between two streamlines is equal to the volume flow rate per unit width between the two streamlines. Thus, if the streamlines are very close together, the speed of the fluid between them is large relative to locations where the same two streamlines are far apart. **Professor Flows noticed a region in which the streamlines were very close together, implying high relative speed in that region of the flow**.

***Discussion*** If the values of $\psi$ on the contour plot are labeled, we can actually infer the fluid speed by measuring the distance between streamlines.

## 9-65

**Solution** For the given set of streamlines, we are to discuss how we can tell the relative speed of the fluid.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is axisymmetric.

***Analysis*** As with 2-D flow, when streamlines that are initially equally spaced spread away from each other, it indicates that the flow speed has decreased in that region. Likewise, if the streamlines come closer together, the flow speed has increased in that region. From the figure provided in the problem statement, we infer that the flow far upstream of the plate is straight and uniform, since the streamlines are parallel. The fluid decelerates as it approaches the front face of the cylinder, especially near the stagnation point, as indicated by the wide gap between streamlines. The flow accelerates rapidly to very high speeds around the corner of the cylinder as indicated by the tightly spaced streamlines there. The flow is seen to separate on top of the cylinder. Since the streamlines are very sparse in this region, **we infer that the fluid moves relatively slowly inside the separation bubble**.

***Discussion*** Such analyses in axisymmetric flow fields are more difficult than those in 2-D planar flow fields because streamlines of equally spaced stream function are *not* spaced equally apart in a uniform axisymmetric flow field. This is due to the fact that the cross-sectional area between streamlines increases with radius (a factor of $2\pi r$ is introduced). Nevertheless, we can still tell where the flow speeds up and slows down in this example.

---

## 9-66E

**Solution** We are to interpret a streamline plot by determining the direction of flow and by estimating the speed of the flow at a point.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible **3** The flow is two-dimensional.

***Analysis*** (*a*) We must tilt our heads nearly upside down to see an increase in stream function $\psi$ in the mathematically positive manner. In other words, since $\psi$ increases in the downward direction, the flow is to the lower left, following our left side rule. Arrows are drawn in Fig. 1.

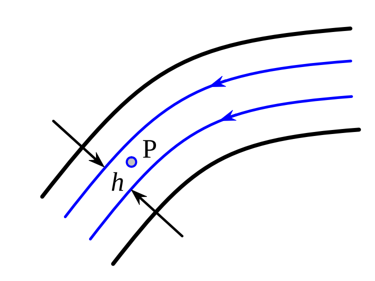



**FIGURE 1**
Streamlines with direction shown.

(*b*) For 2-D incompressible flow the difference in the value of the stream function between two streamlines is equal to the volume flow rate per unit width between the two streamlines. We approximate the flow as uniform between the two labeled streamlines in the figure provided in the problem statement. The speed at point P is thus

$$V_P \approx \frac{\dot{V}}{Wh} = \frac{1}{h}\frac{\dot{V}}{W} = \frac{1}{h}\left(\psi_1 - \psi_2\right) = \frac{1}{2.0\text{ in}}\left(0.45 - 0.32\right)\frac{\text{ft}^2}{\text{s}}\left(\frac{12\text{ in}}{\text{ft}}\right) = \mathbf{0.78}\,\frac{\mathbf{ft}}{\mathbf{s}} \qquad (1)$$

(*c*) Nowhere did we use any property of the fluid, so changing to water does not change our result. **For either air or water (or any incompressible fluid), $V_P$ = 0.78 ft/s**.

***Discussion*** Streamlines and stream functions are *kinematic* properties, as discussed in Chap. 4. That is why fluid density, viscosity, etc. are irrelevant here.

---

## 9-67

**Solution** We are to find the primary dimensions and primary units of the compressible stream function.

***Analysis*** From the given definition, we see that $\psi_\rho$ is the product of a density, a velocity, and a length,

*Primary dimensions of $\psi_\rho$:*

$$\left\{\psi_\rho\right\} = \left\{\frac{\text{mass}}{\text{length}^3}\times\frac{\text{length}}{\text{time}}\times\text{length}\right\} = \left\{\frac{\text{m}}{\text{Lt}}\right\}$$

**The primary units of $\psi_\rho$ are kg/(m·s) (SI) and lbm/(ft s) (English)**.

***Discussion*** Ironically, although the stream function is often applied to potential flows where viscosity is not a parameter, $\psi_\rho$ has the same units as $\mu$.

---

**9-68**

**Solution** We are to generate an expression for the compressible stream function for a given flow field.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** We start by picking one of the two definitions of the compressible stream function (it doesn't matter which part we choose – the solution will be identical).

$$\frac{\partial \psi_\rho}{\partial y} = \rho u = (\rho_1 + C_\rho x)(u_1 + C_u x) = \rho_1 u_1 + (\rho_1 C_u + u_1 C_\rho)x + C_\rho C_u x^2 \quad (1)$$

Next we integrate Eq. 1 with respect to *y*, noting that this is a *partial* integration and we must add an arbitrary function of the other variable, *x*, rather than a simple constant of integration.

$$\psi_\rho = \rho_1 u_1 y + (\rho_1 C_u + u_1 C_\rho)xy + C_\rho C_u x^2 y + g(x) \quad (2)$$

Now we choose the other part of the definition of $\psi$, differentiate Eq. 2, and rearrange as follows:

$$-\rho v = \frac{\partial \psi_\rho}{\partial x} = (\rho_1 C_u + u_1 C_\rho)y + 2C_\rho C_u xy + g'(x) \quad (3)$$

where $g'(x)$ denotes $dg/dx$ since *g* is a function of only one variable, *x*. We now have two expressions for $-\rho v$, Eq. 3 and the value computed from the known density and velocity, i.e.

$$-\rho v = (\rho_1 C_u + u_1 C_\rho)y - 2C_u C_\rho xy \quad (4)$$

We equate Eqs. 3 and 4 and integrate with respect to *x* to find $g(x)$,

$$g'(x) = 0 \qquad g(x) = C \quad (5)$$

Note that here we have added an arbitrary constant of integration *C* since *g* is a function of *x* only. Plugging Eq. 5 into Eq. 2 yields

$$\psi_\rho = \rho_1 u_1 y + (\rho_1 C_u + u_1 C_\rho)xy + C_\rho C_u x^2 y + C \quad (6)$$

We determine constant *C* by setting $\psi_\rho = 0$ at $y = 0$ in Eq. 6, yielding $C = 0$. Thus the final expression for the compressible stream function is

*Compressible stream function*:

$$\boxed{\psi_\rho = \rho_1 u_1 y + (\rho_1 C_u + u_1 C_\rho)xy + C_\rho C_u x^2 y} \quad (7)$$

***Discussion*** You can verify by differentiating $\psi_\rho$ that Eq. 7 yields the correct values of *u* and *v*.

**9-69**

**Solution** We are to generate an expression for the compressible stream function for a given flow field.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** We start by picking one of the two definitions of the compressible stream function (it doesn't matter which part we choose – the solution will be identical).

$$\frac{\partial \psi_\rho}{\partial y} = \rho u = \left(\rho_1 + C_\rho x\right)\left(u_1 + C_u x\right) = \rho_1 u_1 + \left(\rho_1 C_u + u_1 C_\rho\right)x + C_\rho C_u x^2 \quad (1)$$

Next we integrate Eq. 1 with respect to *y*, noting that this is a *partial* integration and we must add an arbitrary function of the other variable, *x*, rather than a simple constant of integration.

$$\psi_\rho = \rho_1 u_1 y + \left(\rho_1 C_u + u_1 C_\rho\right)xy + C_\rho C_u x^2 y + g(x) \quad (2)$$

Now we choose the other part of the definition of $\psi$, differentiate Eq. 2, and rearrange as follows:

$$-\rho v = \frac{\partial \psi_\rho}{\partial x} = \left(\rho_1 C_u + u_1 C_\rho\right)y + 2C_\rho C_u xy + g'(x) \quad (3)$$

where $g'(x)$ denotes $dg/dx$ since *g* is a function of only one variable, *x*. We now have two expressions for $-\rho v$, Eq. 3 and the value computed from the known density and velocity, i.e.

$$-\rho v = \left(\rho_1 C_u + u_1 C_\rho\right)y - 2C_u C_\rho xy \quad (4)$$

We equate Eqs. 3 and 4 and integrate with respect to *x* to find *g*(*x*),

$$g'(x) = 0 \qquad g(x) = C \quad (5)$$

Note that here we have added an arbitrary constant of integration *C* since *g* is a function of *x* only. Plugging Eq. 5 into Eq. 2 yields

$$\psi_\rho = \rho_1 u_1 y + \left(\rho_1 C_u + u_1 C_\rho\right)xy + C_\rho C_u x^2 y + C \quad (6)$$

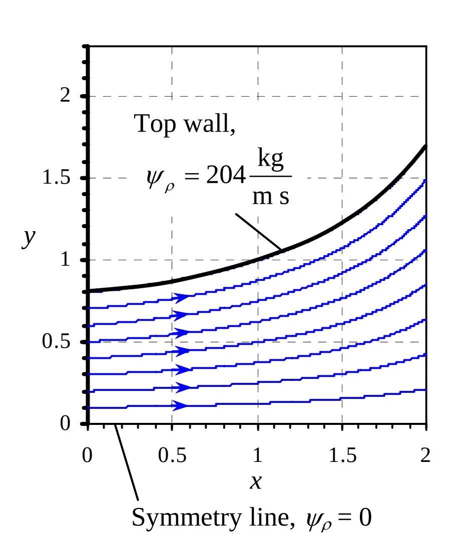



**FIGURE 1**
Streamlines for a diverging duct.

We determine constant *C* by setting $\psi_\rho = 0$ at $y = 0$ in Eq. 6, yielding $C = 0$. Thus the final expression for the compressible stream function is

*Compressible stream function*:
$$\boxed{\psi_\rho = \rho_1 u_1 y + \left(\rho_1 C_u + u_1 C_\rho\right)xy + C_\rho C_u x^2 y} \quad (7)$$

We solve Eq. 7 for *y* as a function of *x* and $\psi_\rho$ so that we can plot streamlines,

*Equation for plotting streamlines*:
$$y = \frac{\psi_\rho}{\rho_1 u_1 + \left(\rho_1 C_u + u_1 C_\rho\right)x + C_\rho C_u x^2} \quad (8)$$

We plot Eq. 8 in Fig. 1 for several values of $\psi_\rho$, using the values of constants $u_1$, $\rho_1$, $C_u$, and $C_\rho$ given in Problem 9-21. The agreement with the streamlines of Problem 9-21 is excellent.

The streamline starting at $x = 0$, $y = 0.8$ m is the top wall of the duct. Therefore the value of $\psi_\rho$ at the top wall of the diverging duct is found be setting at $x = 0$ and $y = 0.8$ m,

*$\psi_\rho$ at the top wall*:
$$\psi_{\rho,\text{top}} = \rho_1 u_1 y = \left(0.85\frac{\text{kg}}{\text{m}^3}\right)\left(300\frac{\text{m}}{\text{s}}\right)(0.8\text{ m}) = \mathbf{204\frac{kg}{m\cdot s}} \quad (9)$$

***Discussion*** You can verify by differentiating $\psi_\rho$ that Eq. 9 yields the correct values of *u* and *v*.

## 9-70

**Solution** We are to interpret a streamline plot by determining the direction of flow and by estimating the speed of the flow at a point.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible **3** The flow is two-dimensional.

***Analysis*** (*a*) We can tell the direction of the flow by whether $\psi_\rho$ increases or decreases in the vertical direction (left side rule). We see that at points A and B the flow is to the right. Furthermore, since the streamlines near point B are somewhat further apart than those near point A (by a factor of about 1.6), the speed at point A is a factor of about 1.6 greater than that at point B. Arrows are drawn in Fig. 1.

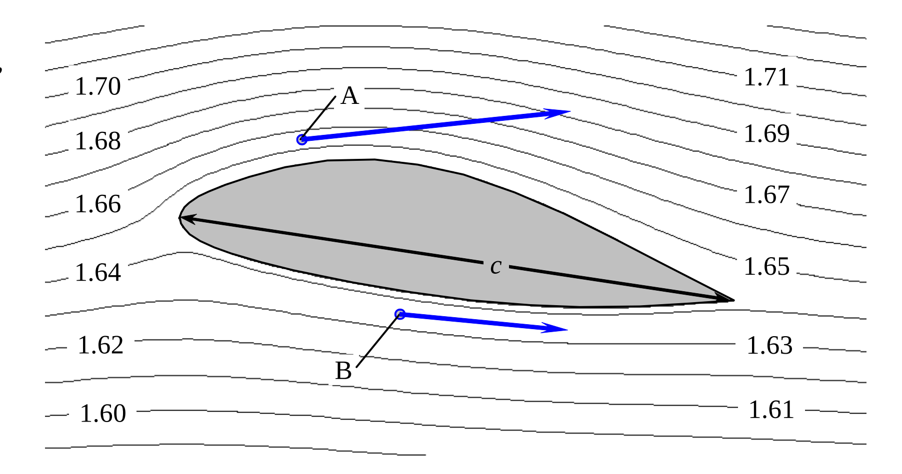



**FIGURE 1**
Relative velocity vectors at points A and B, added to the streamline plot.

In terms of lift, it is obvious that the flow speeds near the upper surface of the hydrofoil are greater than those near the lower surface. From the Bernoulli equation we know that low speeds lead to (relatively) higher pressures; thus the pressure on the lower half of the hydrofoil is greater than that on the upper half, leading to lift.

(*b*) For 2-D incompressible flow the difference in the value of the stream function between two streamlines is equal to the volume flow rate per unit width between the two streamlines. We approximate the flow as uniform between the two streamlines that enclose point A in Fig. P9-70. By measurement with a ruler, we find that the distance $\delta$ between streamlines 1.65 and 1.66 is about 0.034*c*, or about (0.034)(9.0 mm) = 0.306 mm. The speed at point A is thus

$$V_{\text{A}} \approx \frac{\dot{V}}{W\delta} = \frac{1}{\delta}\frac{\dot{V}}{W} = \frac{1}{\delta}\left(\psi_{1.66} - \psi_{1.65}\right)$$
$$= \frac{1}{0.306\times10^{-3}\ \text{m}}\left(1.66-1.65\right)\frac{\text{m}^2}{\text{s}} = 32.7\frac{\text{m}}{\text{s}} \cong \mathbf{33.}\frac{\mathbf{m}}{\mathbf{s}}$$

We give our answer to only two significant digits here because of the difficulty of measuring the distance between the two streamlines.

***Discussion*** Students' answers may vary somewhat depending on how accurately they measure the distance between streamlines. Values between 30 and 40 m/s are reasonable.

**9-71**

**Solution** We are to interpret a streamline plot by determining the direction of flow and by estimating the speed of the flow at a point.

***Assumptions*** **1** The flow is steady (time-averaged). **2** The flow is incompressible **3** The flow is two-dimensional.

***Properties*** The density of air at $T$ = 20°C is 1.18 kg/m³.

***Analysis*** (*a*) We can tell the direction of the flow by whether $\psi_\rho$ increases or decreases in the vertical direction (left side rule). We see that at point A, the flow is to the left, while at point B the flow is to the right. Furthermore, since the streamlines near point B are much closer together than those near point A (by a factor of about five), the speed at point B is a factor of about five greater than that at point A. Arrows are drawn in Fig. 1.

(*b*) For 2-D incompressible flow the difference in the value of the compressible stream function between two streamlines is equal to the mass flow rate per unit width between the two streamlines. We approximate the flow as uniform between the two streamlines that enclose point B in Fig. P9-71. By measurement with a ruler, we find that the distance $\delta$ between streamlines 5 and 6 is about $h$/10, or about 0.10 m. The speed at point B is thus

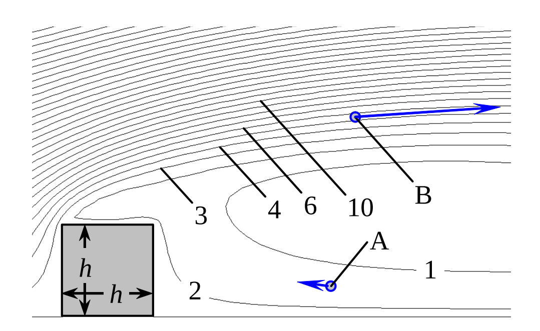



**FIGURE 1**
Relative velocity vectors at points A and B, added to the streamline plot.

$$V_B \approx \frac{\dot{m}}{\rho W \delta} = \frac{1}{\rho\delta}\frac{\dot{m}}{W} = \frac{1}{\rho\delta}\left(\psi_6 - \psi_5\right) = \frac{1}{\left(1.18\,\frac{\text{kg}}{\text{m}^3}\right)\left(0.10\text{ m}\right)}\left(6-5\right)\frac{\text{kg}}{\text{m}\cdot\text{s}} = \mathbf{8.5\,\frac{m}{s}} \quad (1)$$

We are only accurate to one digit here because of the difficulty of measuring the distance between the two streamlines. We give our final result as $\mathbf{V_B}$ **= 8 or 9 m/s**.

***Discussion*** Students' answers may vary considerably depending on how accurately they measure the distance between streamlines.

## Linear Momentum Equations, Boundary Conditions, and Applications

### 9-72C

**Solution** We are to discuss each term, and write the equation as a word equation.

*Analysis* **Term I is the net body force acting on the control volume. Term II is the net surface force acting on the control volume. Term III is the net rate of change of linear momentum within the control volume. Term IV is the net rate of outflow of linear momentum through the control surface**. In words, the equation can be expressed as: "**The total force acting on the control volume is the sum of body forces and surface forces, and is equal to the rate at which momentum changes within the control volume plus the rate at which momentum flows out of the control volume**."

***Discussion*** The dimensions of each term in the equation are those of momentum per time. Each term has primary dimensions of {mLt$^{-2}$}.

### 9-73C

**Solution** We are to discuss velocity boundary conditions in a stationary and a moving frame of reference for the case of an airplane flying through the air.

*Analysis* (*a*) From the stationary frame of reference, $\vec{V} = \vec{V}_{\text{airplane}}$ on all surfaces of the airplane, (no-slip boundary condition). Far from the airplane **the air is still** ($\vec{V}$ **= 0**).

(*b*) From the reference frame moving with the airplane, $\vec{V} = 0$ on all surfaces, (no-slip boundary condition). Far from the airplane the air is moving towards the airplane at **a speed that is opposite the airplane's speed** ($\vec{V} = -\vec{V}_{\text{airplane}}$).

***Discussion*** The no-slip condition requires that the fluid velocity equal the airplane velocity everywhere on the airplane surface, regardless of the geometry of the airplane, and regardless of the frame of reference.

### 9-74C

**Solution** We are to describe the constitutive equations and name the equation to which they are applied.

*Analysis* **The constitutive equations are relationships between the components of the stress tensor and the primary unknowns of the problem, namely pressure and velocity**. The constitutive equations enable us to write the components of the stress tensor in *Cauchy's equation* in terms of the velocity field and the pressure field.

***Discussion*** Cauchy's equation by itself is useless without the constitutive equations, because we would have too many unknowns for the number of available equations.

### 9-75C

**Solution** We are to define mechanical pressure and discuss its application.

*Analysis* **Mechanical pressure is the mean normal stress acting inwardly on a fluid element**. For an incompressible fluid, the density is constant and therefore we have no equation of state available for calculation of the thermodynamic pressure. In fact, thermodynamic pressure cannot even be defined for an incompressible fluid. Fluid elements and surfaces still "feel" a pressure, however, and this pressure is the so-called mechanical pressure.

***Discussion*** When dealing with incompressible fluid flows, pressure variable $P$ is always interpreted as the mechanical pressure $P_m$.

## 9-76C

**Solution** We are to discuss the difference between Newtonian fluids and non-Newtonian fluids, and we are to give examples of each.

***Analysis*** The main distinction between a Newtonian fluid and a non-Newtonian fluid is that **for flow of a Newtonian fluid, shear stress is linearly proportional to shear strain rate**, whereas for flow of a non-Newtonian fluid, the relationship between shear stress and shear strain rate is nonlinear.

There are many examples of Newtonian fluids. Most pure, common liquids like water, oil, gasoline, alcohol, etc. are Newtonian. Most gases also behave like Newtonian fluids. Non-newtonian fluids include paint, pastes and creams, polymer solutions, cake batter, slurries and colloidal suspensions like quicksand, blood, etc.

***Discussion*** The Navier-Stokes equations apply only to Newtonian fluids. For non-Newtonian fluids, you would need to insert nonlinear constitutive equations into Cauchy's equations in order to obtain a useful differential equation for conservation of linear momentum.

---

## 9-77C

**Solution** We are to define or describe each type of fluid.

***Analysis***

(*a*) **A viscoelastic fluid is a fluid that returns (either fully or partially) to its original shape after the applied stress is released**.

(*b*) **A pseudoplastic fluid is a shear thinning fluid – the more the fluid is sheared, the less viscous it becomes**.

(*c*) **A dilatant fluid is a shear thickening fluid – the more the fluid is sheared, the more viscous it becomes**.

(*d*) **A Bingham plastic fluid is an extreme type of pseudoplastic fluid that requires a finite stress called the *yield stress* in order for the fluid to flow at all**.

***Discussion*** All of the above are examples of non-Newtonian fluids.

---

## 9-78

**Solution** We are to generate and discuss velocity and pressure boundary conditions for the given flow problem.

***Assumptions*** **1** The flow is steady in the mean. **2** Surface tension effects are negligibly small.

***Analysis*** On all tank walls, $\vec{V} = 0$ since the tank walls are stationary (no-slip boundary condition). Mathematically, we write **$u_r = u_\theta = u_z = 0$ at $r = R_{\text{tank}}$** (the tank side walls) and at $z = 0$ (the bottom wall of the tank). On the blade surfaces, the fluid velocity must equal that of the blades (also the no-slip condition). At any radial location $r$ the velocity of the blade surface is $\vec{V}_{\text{blade}} = r\omega\vec{e_\theta}$. In other words **$u_\theta = r\omega$ at the blade surfaces**. Since the blades do not move at all in the radial or vertical directions, **$u_r = u_z = 0$ along the blade surfaces**. Finally, **at the free surface $P = P_{\text{atm}}$** since the free surface is exposed to atmospheric air. In addition, the vertical component of velocity **$u_z$ must equal zero at the free surface**. We note that the other two velocity components ($u_r$ and $u_\theta$) may be non-zero at the free surface, but the shear stress in the horizontal plane of the free surface must be zero (negligible shear due to the air). Mathematically, **$\partial u_r/\partial z = \partial u_\theta/\partial z = 0$ at the free surface**.

***Discussion*** The no-slip condition requires that $u_\theta = r\omega$ everywhere on the blade surface, regardless of the geometry of the blades.

---

**9-79**

**Solution** We are to generate and discuss velocity and pressure boundary conditions for the stirrer flow problem from a rotating frame of reference.

***Assumptions*** **1** The flow is steady in the mean. **2** Surface tension effects are negligibly small.

***Analysis*** On all tank walls, $\vec{V}_{\text{tank}} = 0$ from a stationary frame of reference since the tank walls are stationary (no-slip boundary condition). From the *rotating* frame of reference however, the tank walls are rotating in the opposite direction of $\omega$. Mathematically, we write $\boldsymbol{u_r = u_z = 0}$ **and** $\boldsymbol{u_\theta = -R_{\text{tank}}\omega}$ **at** $\boldsymbol{r = R_{\text{tank}}}$ (the tank side walls). **At the bottom wall of the tank we write** $\boldsymbol{u_r = u_z = 0}$ **and** $\boldsymbol{u_\theta = -r\omega}$ **at** $\boldsymbol{z = 0}$. On the blade surfaces, the fluid velocity must equal that of the blades (also the no-slip condition). Since the blades are stationary in this rotating frame of reference, $\boldsymbol{u_r = u_z = u_\theta = 0}$ **at the blade surfaces**. Finally, **at the free surface** $\boldsymbol{P = P_{\text{atm}}}$ since the free surface is exposed to atmospheric air. In addition, the vertical component of velocity $u_z$ must equal zero at the free surface. We note that the other two velocity components ($u_r$ and $u_\theta$) may be non-zero at the free surface, but the shear stress in the horizontal plane of the free surface must be zero (negligible shear due to the air). Mathematically, $\boldsymbol{\partial u_r/\partial z = \partial u_\theta/\partial z = 0}$ **at the free surface**.

***Discussion*** In this problem the free surface boundary conditions are independent of frame of reference.

---

**9-80**

**Solution** We are to generate and discuss velocity and pressure boundary conditions for the given flow problem.

***Assumptions*** **1** The flow is steady. **2** Surface tension effects are negligibly small.

***Analysis*** We must satisfy the no-slip boundary condition on all tank walls, $\vec{V}_{\text{liquid}} = \vec{V}_{\text{tank}}$. Mathematically, we write $\boldsymbol{u_r = u_z = 0}$ **and** $\boldsymbol{u_\theta = R\omega}$ **at** $\boldsymbol{r = R}$ (the tank side walls). We also write $\boldsymbol{u_r = u_z = 0}$ **and** $\boldsymbol{u_\theta = r\omega}$ **at** $\boldsymbol{z = 0}$ (the bottom wall of the tank). We do not specify the pressure along the tank walls. **At the free surface** $\boldsymbol{P = P_{\text{atm}}}$ since the free surface is exposed to atmospheric air. In addition, the vertical and radial components of velocity $\boldsymbol{u_z}$ **and** $\boldsymbol{u_r}$ **must equal zero at the free surface**, but the angular velocity component $u_\theta$ is set to $\boldsymbol{u_\theta = r\omega}$ **at the free surface**. We also know that the shear stress at free surface must be zero (negligible shear due to the air). This boundary condition is not needed however since we already know the velocity field. In fact, the velocity field is known right from the start since we are told that the liquid is in solid body rotation: $u_z = u_r = 0$ and $u_\theta = r\omega$ everywhere.

***Discussion*** This is a degenerate case of the Navier-Stokes equation since the fluid is in solid body rotation. Nevertheless, it is useful to think about the required boundary conditions.

---

**9-81**

**Solution** We are to compare Eqs. 1 and 2 to see if they are the same or not.

***Analysis*** We use the product rule to differentiate Eq. 1,

$$\tau_{r\theta} = \tau_{\theta r} = \mu\left(ru_\theta\frac{-1}{r^2} + \frac{r}{r}\frac{\partial u_\theta}{\partial r} + \frac{1}{r}\frac{\partial u_r}{\partial \theta}\right) = \mu\left(\frac{1}{r}\left(\frac{\partial u_r}{\partial \theta} - u_\theta\right) + \frac{\partial u_\theta}{\partial r}\right) \quad (3)$$

Thus we see that **Eq. 1 and Eq. 2 are equivalent**.

***Discussion*** The viscous stress tensor is defined identically in the other texts; the terms are simply grouped together in a different fashion.

---

**9-82**

**Solution** We are to estimate the volume flow rate of oil between two plates, and we are to calculate the Reynolds number.

***Assumptions*** **1** The flow is steady. **2** The oil is incompressible. **3** Since the gap is so small compared to the plate dimensions, we assume 2-D flow in the *x*-*y* plane. **4** We ignore entrance effects and end effects and assume that the flow can be approximated as fully developed channel flow everywhere in the gap.

***Properties*** The viscosity and density of unused engine oil at $T$ = 60°C are 72.5 × $10^{-3}$ kg/(m·s) and 864 kg/$m^3$ respectively.

***Analysis*** The velocity field for fully developed channel flow is

*Velocity components, 2-D channel flow*:
$$u = \frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right) \qquad v = 0 \tag{1}$$

We integrate the *x* component of velocity times cross-sectional area to obtain volume flow rate (see also Problem 9-44),

*Volume flow rate*:
$$\dot{V} = \int_A u\,dA = \int_{y=0}^{y=h}\frac{1}{2\mu}\frac{dP}{dx}\left(y^2 - hy\right)W\,dy = -\frac{1}{12\mu}\frac{dP}{dx}h^3W \tag{2}$$

The pressure gradient is approximated as

$$\frac{dP}{dx} \approx \frac{P_{\text{out}} - P_{\text{in}}}{L} = \frac{(0-1)\text{ atm}}{1.5\text{ m}}\left(\frac{101{,}300\text{ N/m}^2}{\text{atm}}\right) = -67{,}530\text{ N/m}^3 \tag{3}$$

We plug Eq. 3 into Eq. 2 and solve for the volume flow rate,

*Volume flow rate*:
$$\dot{V} = -\frac{1}{12\left(72.5\times10^{-3}\ \frac{\text{kg}}{\text{m}\cdot\text{s}}\right)}\left(-67{,}530\frac{\text{N}}{\text{m}^3}\right)(0.0025\text{ m})^3(0.75\text{ m})\left(\frac{\text{kg m}}{\text{s}^2\text{N}}\right) \tag{4}$$
$$= 9.0966\times10^{-4}\text{ m}^3/\text{s} \cong \mathbf{9.10\times10^{-4}\ m^3/s}$$

The average velocity of the oil through the channel is

*Average velocity*:
$$V = \frac{\dot{V}}{hW} = \frac{9.0966\times10^{-4}\text{ m}^3/\text{s}}{(0.0025\text{ m})\times(0.75\text{ m})} = 0.48515\text{ m/s} \cong 0.485\text{ m/s} \tag{5}$$

Finally, the characteristic Reynolds number is

$$\text{Re} = \frac{\rho V h}{\mu} = \frac{\left(864\text{ kg/m}^3\right)(0.48515\text{ m/s})(0.0025\text{ m})}{72.5\times10^{-3}\text{ kg/m}\cdot\text{s}} = \mathbf{14.5} \tag{6}$$

**The flow is definitely laminar**.

***Discussion*** We give our final results to three significant digits.

**9-83**

**Solution** For a given velocity field, we are to calculate the pressure field.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane. **4** Gravity does not act in either the *x* or the *y* direction.

***Analysis*** The flow field must satisfy the steady, two-dimensional, incompressible continuity *and* momentum equations. We check each equation separately; let's consider continuity first:

*Continuity*:
$$\underbrace{\frac{\partial u}{\partial x}}_{a} + \underbrace{\frac{\partial v}{\partial y}}_{-a} + \underbrace{\cancel{\frac{\partial w}{\partial z}}}_{0 \text{ (2-D)}} = 0$$

Continuity is satisfied. Now we look at the *x component* of the Navier-Stokes equation:

*x momentum*:
$$\rho\left(\underbrace{\cancel{\frac{\partial u}{\partial t}}}_{0 \text{ (steady)}} + \underbrace{u\frac{\partial u}{\partial x}}_{(ax+b)a} + \underbrace{v\cancel{\frac{\partial u}{\partial y}}}_{(-ay+cx^2)0} + \underbrace{\cancel{w\frac{\partial u}{\partial z}}}_{0 \text{ (2-D)}}\right) = -\frac{\partial P}{\partial x} + \underbrace{\cancel{\rho g_x}}_{0} + \mu\left(\underbrace{\cancel{\frac{\partial^2 u}{\partial x^2}}}_{0} + \underbrace{\cancel{\frac{\partial^2 u}{\partial y^2}}}_{0} + \underbrace{\cancel{\frac{\partial^2 u}{\partial z^2}}}_{0 \text{ (2-D)}}\right) \quad (1)$$

Equation 1 reduces to

*x momentum*:
$$\frac{\partial P}{\partial x} = \rho\left(-a^2 x - ab\right) \quad (2)$$

The *x* momentum equation is satisfied provided we can generate a pressure field that satisfies Eq. 2. In similar fashion we examine the *y* momentum equation,

*y momentum*:
$$\rho\left(\underbrace{\cancel{\frac{\partial v}{\partial t}}}_{0 \text{ (steady)}} + \underbrace{u\frac{\partial v}{\partial x}}_{(ax+b)2cx} + \underbrace{v\frac{\partial v}{\partial y}}_{(-ay+cx^2)(-a)} + \underbrace{\cancel{w\frac{\partial v}{\partial z}}}_{0 \text{ (2-D)}}\right) = -\frac{\partial P}{\partial y} + \underbrace{\cancel{\rho g_y}}_{0} + \mu\left(\underbrace{\frac{\partial^2 v}{\partial x^2}}_{2c} + \underbrace{\cancel{\frac{\partial^2 v}{\partial y^2}}}_{0} + \underbrace{\cancel{\frac{\partial^2 v}{\partial z^2}}}_{0 \text{ (2-D)}}\right)$$

The *y* momentum equation reduces to

*y momentum*:
$$\frac{\partial P}{\partial y} = \rho\left(-acx^2 - 2bcx - a^2 y\right) + 2c\mu \quad (3)$$

The *y* momentum equation is satisfied provided we can generate a pressure field that satisfies Eq. 3.

In the two-dimensional flow under discussion here, the pressure field *P*(*x*,*y*) must be a smooth function of *x* and *y*. Mathematically, this requires that the order of differentiation (*x* then *y* versus *y* then *x*) should not matter. We check whether this is so by differentiating Eqs. 3 and 2 respectively:

*Cross-differentiation*:
$$\frac{\partial^2 P}{\partial y \partial x} = \frac{\partial}{\partial y}\left(\frac{\partial P}{\partial x}\right) = 0 \qquad \frac{\partial^2 P}{\partial x \partial y} = \frac{\partial}{\partial x}\left(\frac{\partial P}{\partial y}\right) = \rho\left(-2acx - 2bc\right) \quad (4)$$

Since the cross-derivative terms in Eq. 4 do not match, *P* is *not* a smooth function of *x* and *y*. Thus, **we are unable to calculate a steady, incompressible, two-dimensional pressure field with the given velocity field**. We cannot proceed any further.

***Discussion*** This problem shows that if a velocity field satisfies the continuity equation (conservation of mass), this does not necessarily guarantee that the velocity field is physically possible. In the present case, for instance, we are unable to find a pressure field that can satisfy the steady form of the Navier-Stokes equation.

---

**9-84**

**Solution** For a given velocity field, we are to calculate the pressure field.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane. **4** Gravity does not act in either the *x* or the *y* direction.

***Analysis*** The flow field must satisfy the steady, two-dimensional, incompressible continuity *and* momentum equations. We check each equation separately; let's consider continuity first:

*Continuity*:
$$\underbrace{\frac{\partial u}{\partial x}}_{-2ax}+\underbrace{\frac{\partial v}{\partial y}}_{2ax}+\underbrace{\cancel{\frac{\partial w}{\partial z}}}_{0\text{ (2-D)}}=0$$

Continuity is satisfied. Now we look at the *x component* of the Navier-Stokes equation:

*x momentum*:
$$\rho\left(\underbrace{\cancel{\frac{\partial u}{\partial t}}}_{0\text{ (steady)}}+\underbrace{u\frac{\partial u}{\partial x}}_{(-ax^2)(-2ax)}+\underbrace{v\cancel{\frac{\partial u}{\partial y}}}_{(2axy)(0)}+\underbrace{w\cancel{\frac{\partial u}{\partial z}}}_{0\text{ (2-D)}}\right)=-\frac{\partial P}{\partial x}+\underbrace{\cancel{\rho g_x}}_{0}+\mu\left(\underbrace{\frac{\partial^2 u}{\partial x^2}}_{-2a}+\underbrace{\cancel{\frac{\partial^2 u}{\partial y^2}}}_{0}+\underbrace{\cancel{\frac{\partial^2 u}{\partial z^2}}}_{0\text{ (2-D)}}\right)\qquad(1)$$

equation 1 reduces to

*x momentum*:
$$\frac{\partial P}{\partial x}=-2\rho a^2x^3-2\mu a\qquad(2)$$

The *x* momentum equation is satisfied provided we can generate a pressure field that satisfies Eq. 2. In similar fashion we examine the *y* momentum equation,

$$y\text{ momentum: }\rho\left(\underbrace{\cancel{\frac{\partial v}{\partial t}}}_{0\text{ (steady)}}+\underbrace{u\frac{\partial v}{\partial x}}_{(-ax^2)(2ay)}+\underbrace{v\frac{\partial v}{\partial y}}_{(2axy)(2ax)}+\underbrace{w\cancel{\frac{\partial v}{\partial z}}}_{0\text{ (2-D)}}\right)=-\frac{\partial P}{\partial y}+\underbrace{\cancel{\rho g_y}}_{0}+\mu\left(\underbrace{\cancel{\frac{\partial^2 v}{\partial x^2}}}_{0}+\underbrace{\cancel{\frac{\partial^2 v}{\partial y^2}}}_{0}+\underbrace{\cancel{\frac{\partial^2 v}{\partial z^2}}}_{0\text{ (2-D)}}\right)$$

The *y* momentum equation reduces to

*y momentum*:
$$\frac{\partial P}{\partial y}=-2\rho a^2x^2y\qquad(3)$$

The *y* momentum equation is satisfied provided we can generate a pressure field that satisfies Eq. 3.

In the two-dimensional flow under discussion here, the pressure field *P*(*x*,*y*) must be a smooth function of *x* and *y*. Mathematically, this requires that the order of differentiation (*x* then *y* versus *y* then *x*) should not matter. We check whether this is so by differentiating Eqs. 3 and 2 respectively:

*Cross-differentiation*:
$$\frac{\partial^2 P}{\partial y\partial x}=\frac{\partial}{\partial y}\left(\frac{\partial P}{\partial x}\right)=0\qquad\frac{\partial^2 P}{\partial x\partial y}=\frac{\partial}{\partial x}\left(\frac{\partial P}{\partial y}\right)=-2\rho a^2x^2\qquad(4)$$

Since the cross-derivative terms in Eq. 4 do not match, *P* is *not* a smooth function of *x* and *y*. Thus, **we are unable to calculate a steady, incompressible, two-dimensional pressure field with the given velocity field**. We cannot proceed any further.

***Discussion*** This problem shows that even if a velocity field satisfies the continuity equation (conservation of mass), and even if we can plot streamlines for the flow field, this does not necessarily guarantee that the velocity field is physically possible. In the present case, for instance, we are unable to find a pressure field that can satisfy the steady Navier-Stokes equation.

**9-85**

**Solution** For a given velocity field, we are to calculate the pressure field.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the $r$-$\theta$ plane. **4** Gravity does not act in either the $r$ or the $\theta$ direction.

***Analysis*** The flow must satisfy the steady, two-dimensional, incompressible continuity *and* momentum equations. We check each equation separately, starting with continuity,

*Continuity*:
$$\underbrace{\frac{1}{r}\frac{\partial(ru_r)}{\partial r}}_{0}+\underbrace{\frac{1}{r}\frac{\partial(u_\theta)}{\partial \theta}}_{0}+\underbrace{\frac{\partial(u_z)}{\partial z}}_{0}=0$$

Continuity is satisfied. Now we look at the $\theta$ component of the Navier-Stokes equation,

$$\rho\left(\underbrace{\frac{\partial u_\theta}{\partial t}}_{0\text{ (steady)}}+\underbrace{u_r\frac{\partial u_\theta}{\partial r}}_{\frac{C}{r}\left(-\frac{K}{r^2}\right)}+\underbrace{\frac{u_\theta}{r}\frac{\partial u_\theta}{\partial \theta}}_{\frac{K}{r^2}(0)}+\underbrace{\frac{u_r u_\theta}{r}}_{\frac{CK}{r^3}}+\underbrace{u_z\frac{\partial u_\theta}{\partial z}}_{0\text{ (2-D)}}\right)\tag{1}$$
$$=-\frac{1}{r}\frac{\partial P}{\partial \theta}+\underbrace{\rho g_\theta}_{0}+\mu\left(\underbrace{\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_\theta}{\partial r}\right)}_{\frac{K}{r^3}}-\underbrace{\frac{u_\theta}{r^2}}_{\frac{K}{r^3}}+\underbrace{\frac{1}{r^2}\frac{\partial^2 u_\theta}{\partial \theta^2}}_{0}-\underbrace{\frac{2}{r^2}\frac{\partial u_r}{\partial \theta}}_{0}+\underbrace{\frac{\partial^2 u_\theta}{\partial z^2}}_{0\text{ (2-D)}}\right)$$

The $\theta$ momentum equation reduces to

*$\theta$ momentum*:
$$\frac{\partial P}{\partial \theta}=0\tag{2}$$

The $\theta$ momentum equation is satisfied provided we can generate a pressure field that satisfies Eq. 2. As a side note, we might have expected Eq. 2 without even working through the algebra, since in this problem the velocity field is independent of angle $\theta$; we expect that pressure does not depend on $\theta$ either. In similar fashion the $r$ momentum equation is

$$\rho\left(\underbrace{\frac{\partial u_r}{\partial t}}_{0\text{ (steady)}}+\underbrace{u_r\frac{\partial u_r}{\partial r}}_{\frac{C}{r}\left(\frac{-C}{r^2}\right)}+\underbrace{\frac{u_\theta}{r}\frac{\partial u_r}{\partial \theta}}_{\frac{K}{r^2}(0)}-\underbrace{\frac{u_\theta^{\,2}}{r}}_{\frac{K^2}{r^3}}+\underbrace{u_z\frac{\partial u_r}{\partial z}}_{0\text{ (2-D)}}\right)$$
$$=-\frac{\partial P}{\partial r}+\underbrace{\rho g_r}_{0}+\mu\left(\underbrace{\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_r}{\partial r}\right)}_{\frac{C}{r^3}}-\underbrace{\frac{u_r}{r^2}}_{\frac{C}{r^3}}+\underbrace{\frac{1}{r^2}\frac{\partial^2 u_r}{\partial \theta^2}}_{0}-\underbrace{\frac{2}{r^2}\frac{\partial u_\theta}{\partial \theta}}_{0}+\underbrace{\frac{\partial^2 u_r}{\partial z^2}}_{0\text{ (2-D)}}\right)$$

which reduces to

*$r$ momentum*:
$$\frac{\partial P}{\partial r}=\rho\frac{K^2+C^2}{r^3}\tag{3}$$

The $r$ momentum equation is satisfied provided we can generate a pressure field that satisfies Eq. 3.

The pressure field $P(r,\theta)$ must be a smooth function of $r$ and $\theta$. Mathematically, this requires that the order of differentiation ($r$ then $\theta$ versus $\theta$ then $r$) should not matter. We therefore check whether this is so by differentiating Eqs. 2 and 3 respectively:

*Cross-differentiation*: $$\frac{\partial^2 P}{\partial r \partial \theta} = \frac{\partial}{\partial r}\left(\frac{\partial P}{\partial \theta}\right) = 0 \qquad \frac{\partial^2 P}{\partial \theta \partial r} = \frac{\partial}{\partial \theta}\left(\frac{\partial P}{\partial r}\right) = 0 \qquad (4)$$

Equation 4 shows that indeed, $P$ is a smooth function of $r$ and $\theta$. Thus, *we should be able to calculate the pressure field*.

To calculate $P(r,\theta)$, we start with either Eq. 2 or Eq. 3 and integrate. We pick Eq. 2, which we can partially integrate (with respect to $\theta$) to obtain an expression for $P(r,\theta)$,

*Pressure field from θ-momentum*: $$P(r,\theta) = 0 + g(r) \qquad (5)$$

Note that we added an arbitrary function of the other variable $r$ rather than a constant of integration since this is a partial integration. We then take the partial derivative of Eq. 5 with respect to $r$ to obtain

$$\frac{\partial P}{\partial r} = g'(r) = \rho \frac{K^2 + C^2}{r^3} \qquad (6)$$

where we have equated our result to Eq. 3 for consistency. We can now integrate Eq. 6 to obtain the function $g(r)$:

$$g(r) = -\frac{1}{2}\rho \frac{K^2 + C^2}{r^2} + C_1 \qquad (7)$$

where $C_1$ is an arbitrary constant of integration. Finally, we plug Eq. 7 into Eq. 5 to obtain our final expression for $P(x,y)$. The result is

*Answer*: $$\boxed{P(r,\theta) = -\frac{1}{2}\rho \frac{K^2 + C^2}{r^2} + C_1} \qquad (8)$$

Thus the pressure field for this flow decreases like $1/r^2$ as we approach the origin. (The origin itself is a singularity point.) This flow field is a simplistic model of a tornado or hurricane, and the low pressure at the center is the "eye of the storm". We note that this flow field is irrotational, and thus Bernoulli's equation can be used instead to calculate the pressure. If we call the pressure $P_\infty$ far away from the origin ($r \to \infty$), where the local velocity approaches zero, Bernoulli's equation shows that at any distance $r$ from the origin,

*Bernoulli equation*: $$P + \frac{1}{2}\rho V^2 = P_\infty \qquad P = P_\infty - \frac{1}{2}\rho \frac{K^2 + C^2}{r^2} \qquad (9)$$

Equation 9 agrees with our solution (Eq. 8) from the full Navier-Stokes equation if we set constant $C_1$ equal to $P_\infty$. A region of rotational flow near the origin would avoid the singularity there, and would yield a more physically realistic model of a real tornado.

***Discussion*** For practice, try to obtain Eq. 8 by starting with Eq. 3 rather than Eq. 2; you should get the same answer.

**9-86**

**Solution** For a given velocity field, we are to calculate the pressure field.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane. **4** Gravity does not act in either the *x* or the *y* plane.

***Analysis*** The flow field must satisfy the steady, two-dimensional, incompressible continuity *and* momentum equations. We check each equation separately; let's consider continuity first:

*Continuity*:
$$\underbrace{\frac{\partial u}{\partial x}}_{a}+\underbrace{\frac{\partial v}{\partial y}}_{-a}+\underbrace{\cancel{\frac{\partial w}{\partial z}}}_{0\text{ (2-D)}}=0$$

*Continuity is satisfied*. Now we look at the *x component* of the Navier-Stokes equation:

*x momentum*:
$$\rho\left(\underbrace{\cancel{\frac{\partial u}{\partial t}}}_{0\text{ (steady)}}+\underbrace{u\frac{\partial u}{\partial x}}_{(ax+b)a}+\underbrace{v\cancel{\frac{\partial u}{\partial y}}}_{(-ay+c)0}+\underbrace{\cancel{w\frac{\partial u}{\partial z}}}_{0\text{ (2-D)}}\right)=-\frac{\partial P}{\partial x}+\underbrace{\cancel{\rho g_x}}_{0}+\mu\left(\underbrace{\cancel{\frac{\partial^2 u}{\partial x^2}}}_{0}+\underbrace{\cancel{\frac{\partial^2 u}{\partial y^2}}}_{0}+\underbrace{\cancel{\frac{\partial^2 u}{\partial z^2}}}_{0\text{ (2-D)}}\right) \quad (1)$$

The *x* momentum equation reduces to

*x momentum*:
$$\frac{\partial P}{\partial x}=\rho\left(-a^2x-ab\right) \quad (2)$$

The *x* momentum equation is satisfied provided we can generate a smooth pressure field that satisfies Eq. 2. In similar fashion (we don't show the details), the *y* momentum equation reduces to

*y momentum*:
$$\frac{\partial P}{\partial y}=\rho\left(-a^2y+ac\right) \quad (3)$$

The *y* momentum equation is satisfied provided we can generate a smooth pressure field that satisfies Eq. 3. The pressure field $P(x,y)$ must be a smooth function of *x* and *y*. Mathematically, this requires that the order of differentiation (*x* then *y* verses *y* then *x*) should not matter. We therefore check whether this is so by differentiating Eqs. 3 and 2 respectively:

*Cross-differentiation*:
$$\frac{\partial^2 P}{\partial x\partial y}=\frac{\partial}{\partial x}\left(\frac{\partial P}{\partial y}\right)=0 \qquad \frac{\partial^2 P}{\partial y\partial x}=\frac{\partial}{\partial y}\left(\frac{\partial P}{\partial x}\right)=0 \quad (4)$$

Equation 4 shows that indeed, *P* is a smooth function of *x* and *y*. Thus, *we should be able to calculate the pressure field*. To calculate $P(x,y)$, we start with either Eq. 2 or Eq. 3 and integrate. We pick Eq. 2, which we can partially integrate (with respect to *x*) to obtain an expression for $P(x,y)$,

*Pressure field from x-momentum*:
$$P(x,y)=\rho\left(-\frac{a^2x^2}{2}-abx\right)+g(y) \quad (5)$$

Note that we added an arbitrary function of the other variable *y* rather than a constant of integration since this is a partial integration. We then take the partial derivative of Eq. 5 with respect to *y* to obtain

$$\frac{\partial P}{\partial y}=g'(y)=\rho\left(-a^2y+ac\right) \quad (6)$$

where we have equated our result to Eq. 3 for consistency. We can now integrate Eq. 6 to obtain the function $g(y)$:

$$g(y)=\rho\left(-\frac{a^2y^2}{2}+acy\right)+C \quad (7)$$

where *C* is an arbitrary constant of integration. Finally, we plug Eq. 7 into Eq. 5 to obtain our final expression for $P(x,y)$. The result is

*Solution:*

$$\boxed{P(x,y)=\rho\left(-\frac{a^2x^2}{2}-\frac{a^2y^2}{2}-abx+acy\right)+C} \qquad (8)$$

***Discussion*** For practice, you should differentiate Eq. 8 with respect to both *x* and *y*, and compare to Eqs. 2 and 3. (This also serves as a check of our algebra.) In addition, try to obtain Eq. 8 by starting with Eq. 3 rather than Eq. 2; you should get the same answer. Pressure is found to within some arbitrary constant *C* since the absolute magnitude of pressure is irrelevant; only pressure *gradients* are important.

**9-87**

**Solution** For a given geometry and set of boundary conditions, we are to calculate the velocity field, and plot the nondimensionalized velocity profile.

***Assumptions*** We number and list the assumptions for clarity:

**1** The walls are infinite in the *y*-*z* plane (*y* is into the page).
**2** The flow is steady, i.e. time derivatives of any quantity are zero.
**3** The flow is parallel (the *x* component of velocity, *u*, is zero everywhere).
**4** The fluid is incompressible and Newtonian, and the flow is laminar.
**5** Pressure *P* = constant everywhere. In other words, there is no applied pressure gradient pushing the flow; the flow establishes itself due to a balance between gravitational forces and viscous forces.
**6** The velocity field is purely two-dimensional, which implies that *v* = 0 and all *y* derivatives are zero.
**7** Gravity acts in the negative *z* direction. We can express this mathematically as $\vec{g} = -g\vec{k}$, or $g_x = g_y = 0$ and $g_z = -g$.

***Analysis*** We obtain the velocity and pressure fields by following the step-by-step procedure for differential fluid flow solutions.

**Step 1** Set up the problem and the geometry. See problem statement.

**Step 2** List assumptions and boundary conditions. We have already listed seven assumptions. The boundary conditions come from the no-slip condition at the walls (1) at *x* = –*h*/2, *u* = *v* = *w* = 0. (2) At *x* = *h*/2, *u* = *v* = *w* = 0.

**Step 3** Write out and simplify the differential equations. We start with the continuity equation in Cartesian coordinates,

*Continuity*:

$$\underbrace{\frac{\partial u}{\partial x}}_{\text{Assumption 3}} + \underbrace{\frac{\partial v}{\partial y}}_{\text{Assumption 6}} + \frac{\partial w}{\partial z} = 0 \qquad \text{or} \qquad \frac{\partial w}{\partial z} = 0 \qquad (1)$$

Equation 1 tells us that *w* is not a function of *z*. In other words, it doesn't matter where we place our origin – the flow is the same at any *z* location. In other words the flow is *fully developed*. Since *w* is not a function of time (Assumption 2), *z* (Eq. 1), or *y* (Assumption 6), we conclude that *w* is at most a function of *x*,

*Result of continuity*:

$$w = w(x) \text{ only} \qquad (2)$$

We now simplify each component of the Navier-Stokes equation as far as possible. Since *u* = *v* = 0 everywhere and gravity does not act in the *x* or *y* directions, the *x* and *y* momentum equations are satisfied exactly (in fact all terms are zero in both equations). The *z* momentum equation reduces to

*z momentum*:

$$\rho\left(\underbrace{\frac{\partial w}{\partial t}}_{\text{Assumption 2}} + \underbrace{u\frac{\partial w}{\partial x}}_{\text{Assumption 3}} + \underbrace{v\frac{\partial w}{\partial y}}_{\text{Assumption 6}} + \underbrace{w\frac{\partial w}{\partial z}}_{\text{Continuity}}\right) = \underbrace{-\frac{\partial P}{\partial z}}_{\text{Assumption 5}} + \underbrace{\rho g_z}_{-\rho g}$$

$$+\mu\left(\frac{\partial^2 w}{\partial x^2} + \underbrace{\frac{\partial^2 w}{\partial y^2}}_{\text{Assumption 6}} + \underbrace{\frac{\partial^2 w}{\partial z^2}}_{\text{continuity}}\right) \qquad \text{or} \qquad \frac{d^2 w}{dx^2} = \frac{\rho g}{\mu} \qquad (3)$$

We have changed from a partial derivative (∂/∂*x*) to a total derivative (d/dx) in Eq. 3 as a direct result of Eq. 2, reducing the PDE to an ODE.

**Step 4** Solve the differential equations. Continuity and *x* and *y* momentum have already been "solved". Equation 3 (*z* momentum) is integrated twice to get

*Integration of z momentum*:

$$w = \frac{\rho g}{2\mu}x^2 + C_1 x + C_2 \qquad (4)$$

**Step 5** We apply boundary conditions (1) and (2) from Step 2 above to obtain constants $C_1$ and $C_2$,

*Boundary condition* (1): $$0 = \frac{\rho g}{8\mu}h^2 - C_1\frac{h}{2} + C_2$$

and

*Boundary condition* (2): $$0 = \frac{\rho g}{8\mu}h^2 + C_1\frac{h}{2} + C_2$$

We solve the above two equations simultaneously to obtain expressions for $C_1$ and $C_2$,

*Constants of integration*: $$C_1 = 0 \qquad C_2 = \frac{-\rho g}{8\mu}h^2$$

Finally, Eq. 4 becomes

*Final result for velocity field*: $$w = \frac{\rho g}{2\mu}\left(x^2 - \left(\frac{h}{2}\right)^2\right) \qquad (5)$$

Since $-h/2 < x < h/2$ everywhere, $w$ is negative everywhere as expected (flow is downward).

**Step 6** Verify the results. You can plug in the velocity field to verify that all the differential equations and boundary conditions are satisfied.

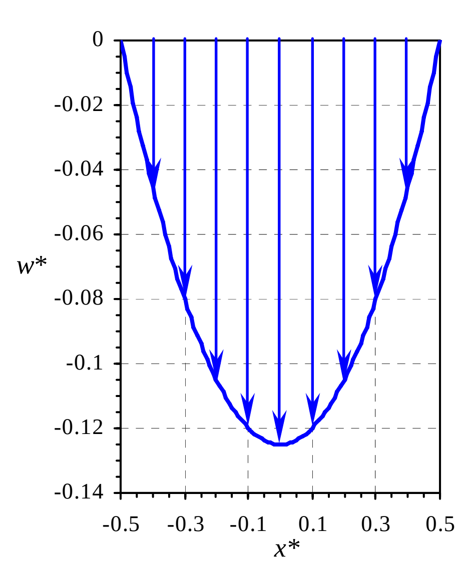



**FIGURE 1**
The velocity profile for liquid falling between two vertical walls.

We nondimensionalize Eq. 5 by inspection: we let $x^* = x/h$ and $w^* = w\mu/(\rho g h^2)$. Eq. 5 becomes

*Nondimensionalized velocity profile*: $$\boxed{w^* = \frac{1}{2}\left(\left(x^*\right)^2 - \frac{1}{4}\right)} \qquad (6)$$

We plot the nondimensional velocity field in Fig. 1. The velocity profile is parabolic.

***Discussion*** Equation 4 for the $z$ component of velocity is identical to that of Example 9-17. In fact, the present problem is identical to Example 9-17 except for the boundary conditions and the location of the origin. Comparing the two results, we see that the maximum nondimensional velocity for the case with two walls is one-fourth than that for the case with only one wall. This is not unexpected – the additional wall leads to more viscous forces that retard the flow.

**9-88**

**Solution** We are to calculate and compare the volume flow rate per unit width of fluid falling between two vertical walls and fluid falling along one vertical wall.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The walls are infinitely wide and very long so that all of the parallel flow, fully developed approximations of the previous problem hold.

***Analysis*** We calculate the volume flow rate per unit width by integration of the velocity:

*Volume flow rate per unit depth, two vertical walls*:

$$\frac{\dot{V}}{L} = \int_{-h/2}^{h/2} w dx = \int_{-h/2}^{h/2} \left[ \frac{\rho g}{2\mu} \left( x^2 - \left( \frac{h}{2} \right)^2 \right) \right] dx = \frac{\rho g}{2\mu} \left[ \frac{x^3}{3} - \frac{h^2}{4} x \right]_{x=-h/2}^{x=h/2} = \frac{\rho g}{2\mu} \left[ \frac{h^3}{24} - \frac{h^3}{8} + \frac{h^3}{24} - \frac{h^3}{8} \right] = \frac{-\rho g h^3}{12\mu} \quad (1)$$

The result is negative since we have defined positive volume flow rate upward, since $z$ is upward, but the flow is downward. For the case with only one vertical wall and a free surface, we calculate the vertical component of velocity to be $w = \frac{\rho g x}{2\mu}(x - 2h)$ (see Example 9-17). Thus, we calculate $\dot{V}/L$ for the case of one vertical wall to be

*Volume flow rate per unit depth, one vertical wall with a free surface*:

$$\frac{\dot{V}}{L} = \int_0^h w dx = \int_0^h \left[ \frac{\rho g x}{2\mu} (x - 2h) \right] dx = \frac{\rho g}{2\mu} \left[ \frac{x^3}{3} - x^2 h \right]_{x=0}^{x=h} = \frac{\rho g}{2\mu} \left[ \frac{h^3}{3} - h^3 - 0 + 0 \right] = \frac{-\rho g h^3}{3\mu} \quad (2)$$

Comparing the two cases we see that $\dot{V}/L$ *for the case of one vertical wall and a free surface is four times greater than the case of two vertical walls with no free surface*. The physical explanation is that with two walls, the fluid is held back by more viscous stresses, leading to a parabolic velocity profile. For the single-wall case the free surface has no shear stress and thus the fluid flows more freely.

***Discussion*** The two flows being compared here are identical except for the boundary conditions. This illustrates the importance of setting proper boundary conditions.

**9-89**

**Solution** For a given geometry and set of boundary conditions, we are to calculate the velocity and pressure fields, and plot the nondimensional velocity profile.

***Assumptions*** We number and list the assumptions for clarity:

1. The wall is infinite in the *s*-*y* plane (*y* is out of the page for a right-handed coordinate system).
2. The flow is steady, i.e. $\frac{\partial}{\partial t}(\text{anything}) = 0$.
3. The flow is parallel and fully developed (we assume the normal component of velocity, $u_n$, is zero, and we assume that the streamwise component of velocity $u_s$ is independent of streamwise coordinate *s*).
4. The fluid is incompressible and Newtonian, and the flow is laminar.
5. Pressure $P$ = constant = $P_{atm}$ at the free surface. In other words, there is no applied pressure gradient pushing the flow; the flow establishes itself due to a balance between gravitational forces and viscous forces along the wall. Atmospheric pressure is constant everywhere since we are neglecting the change of air pressure with elevation.
6. The velocity field is purely two-dimensional, which implies that $v = 0$ and $\frac{\partial}{\partial y}$ (any velocity component) = 0.
7. Gravity acts in the negative *z* direction. We can express this mathematically as $\vec{g} = -g\vec{k}$. In the *s*-*n* plane, $g_s = g\sin\alpha$ and $g_n = -g\cos\alpha$.

***Analysis*** We obtain the velocity and pressure fields by following the step-by-step procedure for differential fluid flow solutions.

**Step 1** Set up the problem and the geometry. See Problem statement.

**Step 2** List assumptions and boundary conditions. We have already listed seven assumptions. The boundary conditions are (1) No slip at the wall: at $n = 0$, $u_s = v = u_n = 0$. (2) At the free surface ($n = h$) there is no shear, which in this coordinate system at the vertical free surface means $\partial u_s/\partial n = 0$. (3) $P = P_{atm}$ at $n = h$.

**Step 3** Write out and simplify the differential equations. We start with the continuity equation in modified Cartesian coordinates, $(s,y,n)$ and $(u_s,v,u_n)$,

*Continuity*:
$$\frac{\partial u_s}{\partial s} + \underbrace{\frac{\partial v}{\partial y}}_{\text{Assumption 6}} + \underbrace{\frac{\partial u_n}{\partial n}}_{\text{Assumption 3}} = 0 \quad \text{or} \quad \frac{\partial u_s}{\partial s} = 0 \qquad (1)$$

Equation 1 tells us that $u_s$ is not a function of *s*. In other words, it doesn't matter where we place our origin – the flow is the same at any *s* location. This does not tell us anything new; we have already assumed that the flow is fully developed (Assumption 3). Furthermore, since $u_s$ is not a function of time (Assumption 2) or *y* (Assumption 6), we conclude that $u_s$ is at most a function of *n*,

*Result of continuity*:
$$u_s = u_s(n) \text{ only} \qquad (2)$$

We now simplify each component of the Navier-Stokes equation as far as possible. Since $v = 0$ everywhere and gravity does not act in the *y* direction, the *y* momentum equation is satisfied exactly (in fact all terms are zero). Since $u_n = 0$ everywhere, the only non-zero terms in the *n* momentum equation are the pressure term and the gravity term. The *n* momentum equation reduces to

*n momentum*:
$$\rho \underbrace{\frac{Du_n}{Dt}}_{\text{Assumption 3}} = -\frac{\partial P}{\partial n} + \underbrace{\rho g_n}_{-\rho g\cos\alpha} + \mu \underbrace{\nabla^2 u_n}_{\text{Assumption 3}} \quad \text{or} \quad \frac{\partial P}{\partial n} = -\rho g\cos\alpha \qquad (3)$$

We integrate Eq. 3 to solve for the pressure,

*Pressure*:
$$P = -\rho g n\cos\alpha + f(s) \qquad (4)$$

where we have added a function of *s* rather than a simple constant of integration. But from boundary condition (3), at $n = h$, $P = P_{atm}$. Thus Eq. 4 yields $f(s) = P_{atm} + \rho g h\cos\alpha$. In other words, $f(s)$ is really not a function of *s* at all. Equation 4 then becomes

*Final expression for pressure*: $$P = P_{\text{atm}} + \rho g(h-n)\cos\alpha \quad (5)$$

The *s* momentum equation reduces to

$$\rho\left(\underbrace{\frac{\partial u_s}{\partial t}}_{\text{Assumption 2}} + \underbrace{u_s\frac{\partial u_s}{\partial s}}_{\text{Continuity}} + \underbrace{v\frac{\partial u_s}{\partial y}}_{\text{Assumption 6}} + \underbrace{u_n\frac{\partial u_s}{\partial n}}_{\text{Assumption 3}}\right) = \underbrace{-\frac{\partial P}{\partial s}}_{\text{Eq. 5}} + \underbrace{\rho g_s}_{\rho g\sin\alpha}$$
$$+\mu\left(\underbrace{\frac{\partial^2 u_s}{\partial s^2}}_{\text{continuity}} + \underbrace{\frac{\partial^2 u_s}{\partial y^2}}_{\text{Assumption 6}} + \frac{\partial^2 u_s}{\partial n^2}\right) \quad \text{or} \quad \frac{d^2u_s}{dn^2} = -\frac{\rho g\sin\alpha}{\mu} \quad (6)$$

We have changed from a partial derivative ($\partial/\partial n$) to a total derivative (d/dn) in Eq. 6 as a direct result of Eq. 2, reducing the PDE to an ODE.

**Step 4** Solve the differential equations. Continuity and *n* and *y* momentum have already been "solved". Equation 6 (*s* momentum) is integrated twice to get

$$u_s = -\frac{\rho g\sin\alpha}{2\mu}n^2 + C_1 n + C_2 \quad (7)$$

**Step 5** We apply boundary conditions (1) and (2) from Step 2 above to obtain constants $C_1$ and $C_2$,

*Boundary condition* (1): $u_s = 0 + 0 + C_2$ at $n = 0$ $\quad C_2 = 0$

and

*Boundary condition* (2):
$$\left.\frac{du_s}{dn}\right)_{n=h} = -\frac{\rho g\sin\alpha}{\mu}h + C_1 = 0 \qquad C_1 = \frac{\rho gh\sin\alpha}{\mu}$$

Finally, Eq. 4 becomes

*Final result for velocity field*: $$u_s = \frac{\rho g\sin\alpha}{2\mu}n(2h-n) \quad (8)$$

Since $n < h$ in the film, $u_s$ is positive everywhere as expected (flow is downward).

**Step 6** Verify the results. You can plug in the velocity field to verify that all the differential equations and boundary conditions are satisfied.

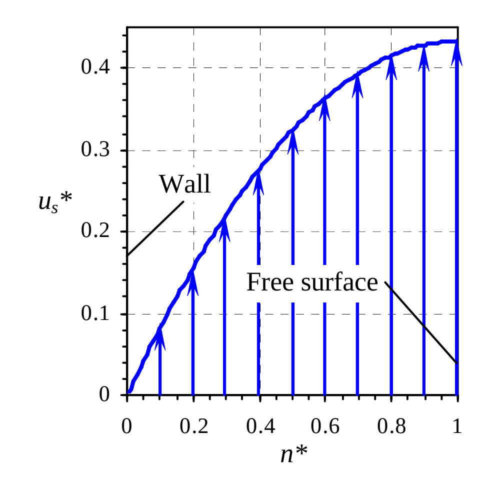



**FIGURE 1**
The velocity profile for an oil film falling down an inclined wall, $\alpha = 60^{\circ}$.

When $\alpha = 90^{\circ}$ $\sin\alpha = 1$ and Eq. 8 is equivalent to Eq. 5 of Example 9-17. (The signs are opposite since *s* is down while *z* is up.) Also, Eq. 5 above reduces to $P = P_{\text{atm}}$ everywhere when $\alpha = 90^{\circ}$ since $\cos\alpha = 0$; this also agrees with the results of Example 9-17. We nondimensionalize Eq. 8 by inspection: we let $n^* = n/h$ and $u_s^* = u_s\mu/(\rho gh^2)$. Eq. 8 becomes

*Nondimensional velocity profile*: $$u_s^* = \frac{n^*}{2}(2-n^*)\sin\alpha \quad (9)$$

We plot the nondimensional velocity field in Fig. 1 for the case in which $\alpha = 60^{\circ}$.

***Discussion*** The profile shape is identical to that of Example 9-17, but is scaled by the factor $\sin\alpha$. This problem could also have been solved in standard Cartesian coordinates $(x,y,z)$, but the algebra would be more involved.

## 9-90

**Solution** We are to calculate the volume flow rate per unit width of oil falling down a vertical wall.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The wall is infinitely wide and very long so that all of the parallel flow, fully developed approximations of Problem 9-89 hold.

***Analysis*** We calculate the volume flow rate per unit width by integration of the velocity:

*Volume flow rate per unit depth*:

$$\frac{\dot{V}}{L} = \int_0^h u_s dn = \int_0^h \left[ \frac{\rho g \sin\alpha}{2\mu} n(2h-n) \right] dn = \frac{\rho g \sin\alpha}{3\mu} h^3 \quad (1)$$

For an oil film of thickness 5.0 mm with $\rho$ = 888 kg/m$^3$ and $\mu$ = 0.80 kg/(m·s), we calculate $\dot{V}/L$ using Eq. 1,

*Result*:

$$\frac{\dot{V}}{L} = \frac{\rho g \sin\alpha}{3\mu} h^3 = \frac{(888 \text{ kg/m}^3)(9.81 \text{ m/s}^2)\sin(60^\circ)(0.005 \text{ m})^3}{3(0.80 \text{ kg/m}\cdot\text{s})} = \mathbf{3.93\times10^{-4}\ m^2/s}$$

***Discussion*** Since viscosity is in the denominator of Eq. 1, a low viscosity liquid (like water) would yield a larger volume flow rate; this agrees with our intuition. Likewise, a larger density liquid and/or a thicker film would yield a larger volume flow rate, again agreeing with our intuition. Finally, if $\alpha = 0^\circ$ there is no flow.

---

## 9-91

**Solution** We are to expand two terms into three terms, and then compress the three terms into one term.

***Analysis*** We use the product rule to differentiate the expression,

$$\mu\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_\theta}{\partial r}\right) - \frac{u_\theta}{r^2}\right) = \mu\left(\frac{\partial^2 u_\theta}{\partial r^2} + \frac{1}{r}\frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r^2}\right)$$

The second part of this question involves some trial and error, using the product rule in reverse. After some effort we get

$$\mu\left(\frac{\partial^2 u_\theta}{\partial r^2} + \frac{1}{r}\frac{\partial u_\theta}{\partial r} - \frac{u_\theta}{r^2}\right) = \mu\left(\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(r u_\theta)\right)\right) \quad (1)$$

You can apply the product rule to verify Eq. 1.

***Discussion*** The grouping of these terms into one term as in Eq. 1 turns out to be useful for some analytical solutions of the Navier-Stokes equation.

---

**9-92**

**Solution** For a given geometry and set of boundary conditions, we are to calculate the velocity field.

***Assumptions*** We number and list the assumptions for clarity:

**1** The cylinders are infinite in the $z$ direction ($z$ is out of the page in the figure of the problem statement for a right-handed coordinate system). The velocity field is purely two-dimensional, which implies that $w$ = 0 and derivatives of any velocity component with respect to $z$ are zero.
**2** The flow is steady, meaning that all time derivatives are zero.
**3** The flow is circular, meaning that the radial velocity component $u_r$ is zero.
**4** The flow is rotationally symmetric, meaning that nothing is a function of $\theta$.
**5** The fluid is incompressible and Newtonian, and the flow is laminar.
**6** Gravitational effects are ignored. (Note that gravity may act in the $z$ direction, leading to an additional hydrostatic pressure distribution in the $z$ direction. This would not affect the present analysis.)

***Analysis*** We obtain the velocity and pressure fields by following the step-by-step procedure for differential fluid flow solutions.

**Step 1** Set up the problem and the geometry. See the problem statement.
**Step 2** List assumptions and boundary conditions. We have already listed five assumptions. The boundary conditions are (1) No slip at the inner wall: at $r = R_i$, $u_\theta = \omega_i R_i$. (2) No slip at the outer wall: at $r = R_o$, $u_\theta = 0$.
**Step 3** Write out and simplify the differential equations. We start with the continuity equation in cylindrical coordinates, $(r,\theta,z)$ and $(u_r,u_\theta,u_z)$,

*Continuity*:
$$\underbrace{\frac{1}{r}\frac{\partial(ru_r)}{\partial r}}_{\text{Assumption 3}} + \underbrace{\frac{1}{r}\frac{\partial(u_\theta)}{\partial\theta}}_{\text{Assumption 4}} + \underbrace{\frac{\partial w}{\partial z}}_{\text{Assumption 1}} = 0 \qquad \text{or} \qquad 0 = 0 \qquad (1)$$

Thus continuity is satisfied exactly by our assumptions.

We now simplify each component of the Navier-Stokes equation as far as possible. Since $w$ = 0 everywhere and gravity is ignored, the $z$ momentum equation is satisfied exactly (in fact all terms are zero). Since $u_r$ = 0 everywhere, the only non-zero terms in the $r$ momentum equation are the pressure term and the "extra" term that involves $u_\theta$. The $r$ momentum equation reduces to

*r momentum*:
$$\frac{\partial P}{\partial r} = \rho\frac{u_\theta^{\,2}}{r} \qquad \text{or} \qquad \frac{dP}{dr} = \rho\frac{u_\theta^{\,2}}{r} \qquad (2)$$

We have changed the partial derivatives to total derivatives since $P$ is a function only of $r$. Equation 2 could be used to solve for $P(r)$ once we find $u_\theta$.

The $\theta$ momentum equation is written out, using the result of the previous problem,

*$\theta$ momentum*:
$$\rho\left(\underbrace{\frac{\partial u_\theta}{\partial t}}_{\text{Assumption 2}} + \underbrace{u_r\frac{\partial u_\theta}{\partial r}}_{\text{Assumption 3}} + \underbrace{\frac{u_\theta}{r}\frac{\partial u_\theta}{\partial\theta}}_{\text{Assumption 4}} + \underbrace{\frac{u_r u_\theta}{r}}_{\text{Assumption 3}} + \underbrace{u_z\frac{\partial u_\theta}{\partial z}}_{\text{Assumption 1}}\right)$$
$$= -\underbrace{\frac{1}{r}\frac{\partial P}{\partial\theta}}_{\text{Assumption 4}} + \underbrace{\rho g_\theta}_{\text{Assumption 6}} + \mu\left(\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}(ru_\theta)\right) + \underbrace{\frac{1}{r^2}\frac{\partial^2 u_\theta}{\partial\theta^2}}_{\text{Assumption 4}} - \underbrace{\frac{2}{r^2}\frac{\partial u_r}{\partial\theta}}_{\text{Assumption 3}} + \underbrace{\frac{\partial^2 u_\theta}{\partial z^2}}_{\text{Assumption 6}}\right)$$

Again we change from partial derivatives ($\partial/\partial r$) to a total derivatives (d/dr), reducing the PDE to an ODE. The $\theta$ momentum equation reduces to

*Reduced $\theta$ momentum*:
$$\frac{d}{dr}\left(\frac{1}{r}\frac{d}{dr}(ru_\theta)\right) = 0 \qquad (3)$$

**Step 4** Solve the differential equations. Continuity and $z$ momentum have already been "solved". Equation 3 ($\theta$ momentum) is integrated once,

*Integration of θ momentum*: $$\frac{1}{r}\frac{d}{dr}\left(ru_\theta\right) = C_1$$

After multiplying by *r* we integrate again. After division by *r* we get

$u_\theta$: $$u_\theta = C_1\frac{r}{2} + \frac{C_2}{r} \qquad (4)$$

**Step 5** We apply boundary conditions (1) and (2) from Step 2 above to obtain constants $C_1$ and $C_2$,

*Boundary condition* (2): $$0 = C_1\frac{R_o}{2} + \frac{C_2}{R_o} \quad \text{or} \quad C_2 = -C_1\frac{R_o^{\,2}}{2}$$

and

*Boundary condition* (1): $$R_i\omega_i = C_1\frac{R_i}{2} + \frac{C_2}{R_i} = C_1\frac{R_i}{2} - C_1\frac{R_o^{\,2}}{2R_i}$$

Which can be solved for $C_1$. The two constants of integration are thus

*Constants of integration*: $$C_1 = \frac{-2R_i^{\,2}\omega_i}{R_o^{\,2} - R_i^{\,2}} \qquad C_2 = \frac{R_o^{\,2}R_i^{\,2}\omega_i}{R_o^{\,2} - R_i^{\,2}}$$

Finally, Eq. 4 becomes (after a bit of algebra)

*Final result for velocity field*: $$\boxed{u_\theta = \frac{R_i^{\,2}\omega_i}{R_o^{\,2} - R_i^{\,2}}\left(\frac{R_o^{\,2}}{r} - r\right)} \qquad (5)$$

**Step 6** Verify the results. You can plug in the velocity field to verify that all the differential equations and boundary conditions are satisfied.

***Discussion*** There are valid alternative forms of Eq. 5. We could integrate Eq. 2 to solve for the pressure since we now know $u_\theta$ from Eq. 5. The algebra is laborious, but not difficult.

**9-93**

**Solution** We are to simplify the velocity field for two limiting cases of Problem 9-92 and discuss.

***Assumptions*** The same assumptions of Problem 9-92 apply here.

***Analysis*** (*a*) First we re-write the velocity profile from Problem 9-92,

*Exact velocity profile*:

$$u_\theta = \frac{R_i^2\omega_i}{R_o^2 - R_i^2}\left(\frac{R_o^2}{r} - r\right) = \frac{R_i^2\omega_i}{(R_o - R_i)(R_o + R_i)}\left(\frac{(R_o - r)(R_o + r)}{r}\right) \qquad (1)$$

Note that Eq. 1 is still exact. When the gap is very small, ($R_o$ – $R_i$) << $R_o$, and $R_o \approx R_i$. Thus we replace $R_o + R_i$ in the denominator of Eq. 1 by $2R_i$. Similarly, $r \approx R_i$ and we replace $R_o + r$ in the numerator of Eq. 1 by $2R_i$. Likewise we replace $r$ in the denominator of Eq. 1 by $R_i$. As suggested we define $y = R_o - r$, $h$ = gap thickness = $R_o - R_i$, and $V$ = speed of the "upper plate" = $R_i\omega_i$ (Fig. 1). Plugging all of these approximations and definitions into Eq. 1 we get

*Approximate velocity for small gap*:

$$u_\theta \approx \frac{R_i^2\omega_i}{h \cdot 2R_i}\left(\frac{y \cdot 2R_i}{R_i}\right) = \frac{y\omega_i R_i}{h} = V\frac{y}{h} \qquad (2)$$

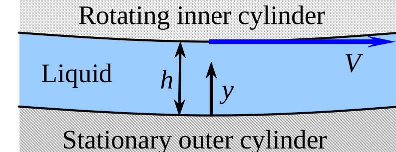



**FIGURE 1**
A magnified view near the bottom for the case in which the gap between the two cylinders is very small.

We verify that Eq. 2 is linear in the small gap and is the same velocity profile as we generated for 2-D Couette flow between two infinite flat plates.

(*b*) As the outer cylinder radius approaches infinity, $R_i$ << $R_o$, and $R_i$ can be ignored when added to or subtracted from $R_o$. Similarly, $r$ << $R_o$, and $r$ can be ignored when added to or subtracted from $R_o$. Equation 1 becomes

*Approximate velocity for infinite* $R_o$:

$$u_\theta \approx \frac{R_i^2\omega_i}{(R_o)(R_o)}\left(\frac{(R_o)(R_o)}{r}\right) = \frac{R_i^2\omega_i}{r} \qquad (3)$$

We recognize Eq. 3 as of the form $u_\theta$ = constant/$r$ which is the velocity field for a **line vortex**.

***Discussion*** Imagine a long, thin cylinder spinning in a vat of liquid. After a long time, the flow field given by Eq. 3 would emerge – basically a line vortex for all radii greater than $R_i$.

## 9-94

**Solution** For a given geometry and set of boundary conditions, we are to calculate the velocity field.

***Assumptions*** The assumptions are identical to those of Problem 9-92.

***Analysis*** We obtain the velocity and pressure fields by following the step-by-step procedure for differential fluid flow solutions. Everything is identical to Problem 9-92 except for the boundary condition at the outer cylinder wall. We re-write boundary condition (2): at $r = R_o$, $u_\theta = \omega_o R_o$. We will not repeat all the algebra associated with the equations of motion. The tangential velocity component is still

$u_\theta$:
$$u_\theta = C_1\frac{r}{2} + \frac{C_2}{r} \qquad (1)$$

Now we apply boundary conditions (1) and (2) to obtain constants $C_1$ and $C_2$,

*Boundary condition* (1):
$$\frac{R_i}{2}C_1 + \frac{1}{R_i}C_2 = R_i\omega_i \qquad (2)$$

*Boundary condition* (2):
$$\frac{R_o}{2}C_1 + \frac{1}{R_o}C_2 = R_o\omega_o \qquad (3)$$

We solve Eqs. 2 and 3 simultaneously for $C_1$ and $C_2$. The result is

*Constants of integration*:
$$C_1 = \frac{2\left(R_o^2\omega_o - R_i^2\omega_i\right)}{R_o^2 - R_i^2} \qquad C_2 = \frac{R_o^2 R_i^2\left(\omega_i - \omega_o\right)}{R_o^2 - R_i^2} \qquad (4)$$

Finally, Eq. 4 becomes (after a bit of algebra)

*Final result for velocity field*:
$$\boxed{u_\theta = \frac{1}{R_o^2 - R_i^2}\left[\left(R_o^2\omega_o - R_i^2\omega_i\right)r + \frac{R_o^2 R_i^2\left(\omega_i - \omega_o\right)}{r}\right]} \qquad (5)$$

We set $\Omega_o = 0$ in Eq. 5 to verify that it simplifies to the result of Problem 9-92,

*Simplified velocity field*:
$$\boxed{u_\theta = \frac{R_i^2\omega_i}{R_o^2 - R_i^2}\left(\frac{R_o^2}{r} - r\right)} \qquad (6)$$

***Discussion*** There are valid alternative forms of Eq. 5.

---

## 9-95

**Solution** We are to discuss a simplified version of the velocity field of the previous problem.

***Assumptions*** The assumptions are identical to those of the previous problem.

***Analysis*** We set $R_i = \omega_i = 0$ in Eq. 5 of the previous problem. The tangential velocity component simplifies to

*Simplified* $u_\theta$:
$$u_\theta = \frac{1}{R_o^2}\left[R_o^2\omega_o r\right] = \omega_o r \qquad (1)$$

We recognize Eq. 1 as the velocity field for *solid body rotation*. To set up this velocity field in a physical experiment, we would place a cylindrical container of liquid on a rotating table. After a long time, the entire tank, including the liquid, would be in solid body rotation.

***Discussion*** If you imagine flow between the inner and outer cylinders, and then imagine that the inner cylinder stops spinning and shrinks to infinitesimal radius, you can convince yourself that solid body rotation would result.

---

**9-96**

**Solution** For flow in a pipe annulus we are to calculate the velocity field.

***Assumptions*** We number and list the assumptions for clarity:

**1** The pipe is infinitely long in the *x* direction.

**2** The flow is steady, i.e. $\frac{\partial}{\partial t}(\text{anything}) = 0$.

**3** This is a parallel flow (the *r* component of velocity, $u_r$, is zero).

**4** The fluid is incompressible and Newtonian, and the flow is laminar.

**5** A constant pressure gradient is applied in the *x* direction such that pressure changes linearly with respect to *x* according to the given expression.

**6** The velocity field is axisymmetric with no swirl, implying that $u_\theta = 0$ and $\frac{\partial}{\partial \theta}(\text{anything}) = 0$.

**7** We ignore the effects of gravity.

***Analysis*** We obtain the velocity field by following the step-by-step procedure for differential fluid flow solutions.

**Step 1** Lay out the problem and the geometry. See the problem statement.

**Step 2** List assumptions and boundary conditions. We have already listed seven assumptions. The boundary conditions come from imposing the no slip condition at both pipe walls: (1) at $r = R_i$, $\vec{V} = 0$. (2) at $r = R_o$, $\vec{V} = 0$.

**Step 3** Write out and simplify the differential equations. We start with the continuity equation in cylindrical coordinates,

*Continuity*:
$$\underbrace{\frac{1}{r}\frac{\partial (r u_r)}{\partial r}}_{\text{Assumption 3}} + \underbrace{\frac{1}{r}\frac{\partial (u_\theta)}{\partial \theta}}_{\text{Assumption 6}} + \frac{\partial u}{\partial x} = 0 \qquad \text{or} \qquad \frac{\partial u}{\partial x} = 0 \qquad (1)$$

Equation 1 tells us that *u* is not a function of *x*. In other words, it doesn't matter where we place our origin – the flow is the same at any *x* location. This can also be inferred directly from Assumption 1, which tells us that there is nothing special about any *x* location since the pipe is infinite in length – the flow is fully developed. Furthermore, since *u* is not a function of time (Assumption 2) or $\theta$ (Assumption 6), we conclude that *u* is at most a function of *r*,

*Result of continuity*:
$$u = u(r) \text{ only} \qquad (2)$$

Next we simplify the *x* momentum equation as far as possible:

*x momentum*:
$$\rho\left(\underbrace{\frac{\partial u}{\partial t}}_{\text{Assumption 2}} + \underbrace{u_r\frac{\partial u}{\partial r}}_{\text{Assumption 3}} + \underbrace{\frac{u_\theta}{r}\frac{\partial u}{\partial \theta}}_{\text{Assumption 6}} + \underbrace{u\frac{\partial u}{\partial x}}_{\text{Continuity}}\right)$$
$$= -\frac{\partial P}{\partial x} + \underbrace{\rho g_x}_{\text{Assumption 7}} + \mu\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u}{\partial r}\right) + \underbrace{\frac{1}{r^2}\frac{\partial^2 u}{\partial \theta^2}}_{\text{Assumption 6}} + \underbrace{\frac{\partial^2 u}{\partial x^2}}_{\text{Continuity}}\right) \qquad (3)$$

or

*Result of x momentum*:
$$\frac{1}{r}\frac{d}{dr}\left(r\frac{du}{dr}\right) = \frac{1}{\mu}\frac{\partial P}{\partial x} \qquad (4)$$

Note that we have replaced the partial derivative operators for the *u* derivatives with total derivative operators because of Eq. 2. Every term in the *r* momentum equation is zero except the pressure gradient term, forcing that lone term to also be zero,

*r momentum*:
$$\frac{\partial P}{\partial r} = 0 \qquad (5)$$

In other words, *P* is not a function of *r*. Since *P* is also not a function of time (Assumption 2) or $\theta$ (Assumption 6), *P* can be at most a function of *x*,

*Result of r momentum*: $$P = P(x) \text{ only} \quad (6)$$

Therefore we can replace the partial derivative operator for the pressure gradient in Eq. 4 by the total derivative operator since $P$ varies only with $x$. Finally, all terms of the $\theta$ component of the Navier-Stokes equation go to zero.

**Step 4** Solve the differential equations. Continuity and $r$ momentum have already been "solved", resulting in Eqs. 2 and 6 respectively. The $\theta$ momentum equation has vanished, and thus we are left with Eq. 4 ($x$ momentum). After multiplying both sides by $r$, we integrate once to obtain

*Integration of x momentum*: $$r\frac{du}{dr} = \frac{r^2}{2\mu}\frac{dP}{dx} + C_1 \quad (7)$$

where $C_1$ is a constant of integration. Note that the pressure gradient $dP/dx$ is a constant here. After dividing both sides of Eq. 7 by $r$, we can integrate a second time to get

*Second integration of x momentum*: $$u = \frac{r^2}{4\mu}\frac{dP}{dx} + C_1 \ln r + C_2 \quad (8)$$

where $C_2$ is a second constant of integration.

**Step 5** Apply boundary conditions from Step 2 above to obtain constants $C_1$ and $C_2$:

*Boundary condition* (1): $$0 = \frac{R_i^{\,2}}{4\mu}\frac{dP}{dx} + C_1 \ln R_i + C_2$$

*Boundary condition* (2): $$0 = \frac{R_o^{\,2}}{4\mu}\frac{dP}{dx} + C_1 \ln R_o + C_2$$

We solve the above two equations simultaneously to find $C_1$ and $C_2$,

*Constants*: $$C_1 = -\frac{\left(R_o^{\,2} - R_i^{\,2}\right)}{4\mu \ln \frac{R_o}{R_i}}\frac{dP}{dx} \qquad C_2 = \frac{\left(R_o^{\,2}\ln R_i - R_i^{\,2}\ln R_o\right)}{4\mu \ln \frac{R_o}{R_i}}\frac{dP}{dx}$$

After some algebra and rearrangement, Eq. 7 becomes

*Final result for axial velocity*: $$\boxed{u = \frac{1}{4\mu}\frac{dP}{dx}\left(r^2 + \frac{R_i^{\,2}\ln \frac{r}{R_o} - R_o^{\,2}\ln \frac{r}{R_i}}{\ln \frac{R_o}{R_i}}\right)} \quad (9)$$

**Step 6** Verify the results. You can plug in the velocity field to verify that all the differential equations and boundary conditions are satisfied.

***Discussion*** There are other valid forms of Eq. 9. For example, after some rearrangement, Eq. 9 can be written as

*Alternative form*: $$\boxed{u = \frac{1}{4\mu}\frac{dP}{dx}\left(r^2 - R_o^{\,2} - \frac{R_o^{\,2} - R_i^{\,2}}{\ln \frac{R_o}{R_i}}\ln \frac{r}{R_o}\right)} \quad (10)$$

---

**9-97**

**Solution** We are to generate the velocity field for a given flow setup.

***Assumptions*** All assumptions are the same as those of the previous problem except for the fifth one, which we modify here: **5** Pressure $P$ is constant everywhere.

***Analysis*** Most of the algebra is identical to that of the previous problem except that the pressure gradient is zero, making this problem easier. Also, the first boundary condition changes: at $r = R_i$, $u = V$. The $x$ momentum equation reduces to

*Result of x momentum*:
$$\frac{1}{r}\frac{d}{dr}\left(r\frac{du}{dr}\right) = 0 \qquad (1)$$

After integration, division by $r$, and a second integration, Eq. 1 yields

*x component of velocity*:
$$u = C_1 \ln r + C_2 \qquad (2)$$

We apply boundary conditions:

*Boundary condition* (1):
$$V = C_1 \ln R_i + C_2$$

and

*Boundary condition* (2):
$$0 = C_1 \ln R_o + C_2$$

We solve the above two equations simultaneously to yield the constants,

*Constants of integration*:
$$C_1 = \frac{-V}{\ln\frac{R_o}{R_i}} \qquad C_2 = \frac{V \ln R_o}{\ln\frac{R_o}{R_i}} \qquad (3)$$

and thus Eq. 2 becomes

*Result for u*:
$$\boxed{u = \frac{V}{\ln\frac{R_o}{R_i}}\left(\ln R_o - \ln r\right) = \frac{V \ln\frac{R_o}{r}}{\ln\frac{R_o}{R_i}}} \qquad (4)$$

***Discussion*** In this and other parallel flow problems, the nonlinear terms in the Navier-Stokes equation drop out, simplifying the problem and enabling an exact analytical solution to be found.

**9-98**

**Solution** We are to generate the velocity field for a given flow setup.

***Assumptions*** All assumptions are the same as those of the previous problem.

***Analysis*** The algebra is identical to that of the previous problem except that the boundary conditions are swapped: at $r = R_i$, $u = 0$ and at $r = R_o$, $u = V$. The $x$ momentum equation reduces to

*Result of x momentum*:
$$\frac{1}{r}\frac{d}{dr}\left(r\frac{du}{dr}\right) = 0 \qquad (1)$$

After integration, division by $r$, and a second integration, Eq. 1 yields

*x component of velocity*:
$$u = C_1 \ln r + C_2 \qquad (2)$$

We apply boundary conditions:

*Boundary condition* (1):
$$V = C_1 \ln R_o + C_2$$

and

*Boundary condition* (2):
$$0 = C_1 \ln R_i + C_2$$

We solve the above two equations simultaneously to yield the constants,

*Constants of integration*:
$$C_1 = \frac{-V}{\ln\frac{R_i}{R_o}} \qquad C_2 = \frac{V \ln R_i}{\ln\frac{R_i}{R_o}} \qquad (3)$$

and thus Eq. 2 becomes

*Result for u*:
$$\boxed{u = \frac{V}{\ln\frac{R_i}{R_o}}\left(\ln R_i - \ln r\right) = \frac{V \ln\frac{R_i}{r}}{\ln\frac{R_i}{R_o}}} \qquad (4)$$

***Discussion*** Since the boundary conditions of the present problem are the same as those of the previous problem except that $R_o$ and $R_i$ are swapped, it turns out that the result is also identical except that the two radii are swapped.

**9-99**

**Solution** For modified Couette flow with two immiscible fluids we are to list the boundary conditions and then solve for both the velocity and pressure fields. Finally we are to plot the velocity profile across the channel.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x*-*y* plane.

***Properties*** The density and viscosity of water at $T = 80^{\circ}$C are 971.8 kg/m$^3$ and $0.355 \times 10^{-3}$ kg/(m·s) respectively. The density and viscosity of unused engine oil at $T = 80^{\circ}$C are 852 kg/m$^3$ and $32.0 \times 10^{-3}$ kg/(m·s) respectively.

***Analysis*** (*a*) The velocity boundary conditions come from the no-slip condition at the walls:

*Boundary condition* (1): At $z = 0$, $u_1 = 0$ (1)

and

*Boundary condition* (2): At $z = h_1 + h_2$, $u_2 = V$ (2)

At the interface we know that both the velocities and the shear stresses must match,

*Boundary condition* (3): At $z = h_1$, $u_1 = u_2$ (3)

and

*Boundary condition* (4): At $z = h_1$, $\mu_1 \frac{du_1}{dz} = \mu_2 \frac{du_2}{dz}$ (4)

The first pressure boundary condition comes from the known pressure on the bottom,

*Boundary condition* (5): At $z = 0$, $P = P_0$ (5)

The second pressure boundary condition comes from the fact that the pressure cannot have a discontinuity at the interface since we are ignoring surface tension,

*Boundary condition* (6): At $z = h_1$, $P_1 = P_2$ (6)

(*b*) We solve for the velocity field using the step-by-step procedure outlined in this chapter. However, we leave out the details because the algebra is identical to that of simple Couette flow – the only difference is in the boundary conditions. For parallel, fully developed flow in the *x* direction, *u* is the only non-zero velocity component and it is a function of *z* only. The *x* momentum equations in the two fluids reduce to

*x momentum*: $$\frac{d^2u_1}{dz^2} = 0 \qquad \frac{d^2u_2}{dz^2} = 0 \qquad (7)$$

We integrate both parts of Eq. 7 twice, introducing four constants of integration,

*Expressions for u*: $$u_1 = C_1 z + C_2 \qquad u_2 = C_3 z + C_4 \qquad (8)$$

We apply the first four boundary conditions to find these constants,

*Boundary conditions* (1) and (2): $$C_2 = 0 \qquad V = C_3\left(h_1 + h_2\right) + C_4$$

and

*Boundary conditions* (3) and (4): $$C_1 h_1 = C_3 h_1 + C_4 \qquad \mu_1 C_1 = \mu_2 C_3$$

After some algebra, we solve simultaneously for all the constants,

$$C_1 = \frac{\mu_2 V}{\mu_2 h_1 + \mu_1 h_2} \qquad C_2 = 0 \qquad C_3 = \frac{\mu_1 V}{\mu_2 h_1 + \mu_1 h_2} \qquad C_4 = V\left(\frac{\mu_2 h_1 - \mu_1 h_1}{\mu_2 h_1 + \mu_1 h_2}\right) \qquad (9)$$

And the velocity components of Eq. 8 become

$$\boxed{u_1 = \frac{\mu_2 V}{\mu_2 h_1 + \mu_1 h_2} z} \qquad (10)$$

and

$$\boxed{u_2 = \frac{\mu_1 V}{\mu_2 h_1 + \mu_1 h_2} z + V\left(\frac{\mu_2 h_1 - \mu_1 h_1}{\mu_2 h_1 + \mu_1 h_2}\right) = \frac{V}{\mu_2 h_1 + \mu_1 h_2}\left(\mu_1 (z - h_1) + \mu_2 h_1\right)} \qquad (11)$$

You should plug in the boundary conditions to verify that Eqs. 10 and 11 are correct.

(*c*) We analyze the *z* momentum equation to find the pressure. Since $w$ = 0 everywhere, the only non-zero terms are the pressure and gravity terms. Thus we have

*z momentum*: $$\frac{dP_1}{dz} = -\rho_1 g \qquad \frac{dP_2}{dz} = -\rho_2 g \qquad (12)$$

We integrate Eqs. 12 to obtain

*Pressure*: $$P_1 = -\rho_1 g z + C_5 \qquad P_2 = -\rho_2 g z + C_6 \qquad (13)$$

After applying boundary conditions (5) and (6) we obtain the final expressions for the two pressures,

$$\boxed{P_1 = P_0 - \rho_1 g z} \text{ and } \boxed{P_2 = P_0 + (\rho_1 + \rho_2) g h_1 - \rho_2 g z} \qquad (14)$$

Again you can verify that the boundary conditions are satisfied by Eq. 14.

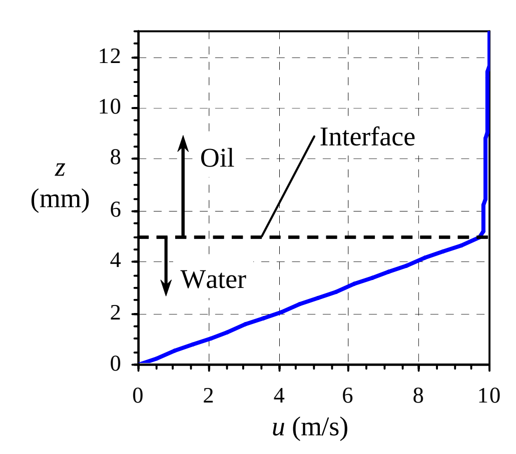



**FIGURE 1**
The velocity profile for Couette flow with two immiscible liquids.

(*d*) For the given fluid properties we plot the velocity profile in Fig. 1. Since the oil is so much more viscous than the water, the oil velocity is nearly constant (small slope) while the water velocity varies rapidly (large slope). At the interface the viscosity times the slope must match, so this should not be surprising.

***Discussion*** Both velocity profiles are linear. The pressure is simply hydrostatic since *P* is a function of *z* only. The oil must be on top since it is less dense than water.

**9-100**

**Solution** We are to calculate $u(r)$ for flow inside an inclined round pipe.

***Assumptions*** We number and list the assumptions for clarity:

**1** The pipe is infinitely long in the $x$ direction.
**2** The flow is steady, i.e. any time derivative is zero.
**3** This is a parallel flow (the $r$ component of velocity, $u_r$, is zero).
**4** The fluid is incompressible and Newtonian, and the flow is laminar.
**5** The pressure is constant everywhere except for hydrostatic pressure.
**6** The velocity field is axisymmetric with no swirl, implying that $u_\theta = 0$ and all derivatives with respect to $\theta$ are zero.

***Analysis*** To obtain the velocity and pressure fields, we follow the step-by-step procedure outlined above.

**Step 1** Lay out the problem and the geometry. See the problem statement.

**Step 2** List assumptions and boundary conditions. We have already listed six assumptions. The first boundary condition comes from imposing the no slip condition at the pipe wall: (1) at $r = R$, $\vec{V} = 0$. The second boundary condition comes from the fact that the centerline of the pipe is an axis of symmetry: (2) at $r = 0$, $du/dr = 0$.

**Step 3** Write out and simplify the differential equations. We start with the continuity equation in cylindrical coordinates, a modified version of Eq. 9-62a,

*Continuity*:
$$\underbrace{\frac{1}{r}\frac{\partial (r u_r)}{\partial r}}_{\text{Assumption 3}} + \underbrace{\frac{1}{r}\frac{\partial (u_\theta)}{\partial \theta}}_{\text{Assumption 6}} + \frac{\partial u}{\partial x} = 0 \qquad \text{or} \qquad \frac{\partial u}{\partial x} = 0 \qquad (1)$$

Equation 1 tells us that $u$ is not a function of $x$. In other words, it doesn't matter where we place our origin – the flow is the same at any $x$ location. This can also be inferred directly from Assumption 1, which tells us that there is nothing special about any $x$ location since the pipe is infinite in length – the flow is fully developed. Furthermore, since $u$ is not a function of time (Assumption 2) or $\theta$ (Assumption 6), we conclude that $u$ is at most a function of $r$,

*Result of continuity*:
$$u = u(r) \text{ only} \qquad (2)$$

We now simplify the $x$ momentum equation as far as possible:

*x momentum*:
$$\rho\left(\underbrace{\frac{\partial u}{\partial t}}_{\text{Assumption 2}} + \underbrace{u_r\frac{\partial u}{\partial r}}_{\text{Assumption 3}} + \underbrace{\frac{u_\theta}{r}\frac{\partial u}{\partial \theta}}_{\text{Assumption 6}} + \underbrace{u\frac{\partial u}{\partial x}}_{\text{Continuity}}\right)$$
$$= -\underbrace{\frac{\partial P}{\partial x}}_{\text{Assumption 5}} + \underbrace{\rho g_x}_{\rho g \sin\alpha} + \mu\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u}{\partial r}\right) + \underbrace{\frac{1}{r^2}\frac{\partial^2 u}{\partial \theta^2}}_{\text{Assumption 6}} + \underbrace{\frac{\partial^2 u}{\partial x^2}}_{\text{Continuity}}\right)$$

or

*Result of x momentum*:
$$\frac{1}{r}\frac{d}{dr}\left(r\frac{du}{dr}\right) = \frac{-\rho g \sin\alpha}{\mu} \qquad (3)$$

As in previous examples the material acceleration (entire left hand side of the $x$ momentum equation) is zero, implying that fluid particles are not accelerating at all in this flow field, and linearizing the Navier-Stokes equation. Also notice that we have replaced the partial derivative operators for the $u$ derivatives with total derivative operators because of Eq. 2.

You can show in similar fashion that every term in the $r$ momentum equation and in the $\theta$ momentum equation goes to zero.

**Step 4** Solve the differential equations. Continuity, $r$ momentum, and $\theta$ momentum have already been solved, and thus we are left with Eq. 3 ($x$ momentum). After multiplying both sides by $r$, integrating, dividing by $r$, and integrating again,

*Axial velocity component*: $$u = \frac{-\rho g \sin\alpha}{4\mu} r^2 + C_1 \ln r + C_2 \quad (4)$$

where $C_1$ and $C_2$ are constants of integration.

**Step 5** Apply boundary conditions from Step 2 above to obtain constants $C_1$ and $C_2$. We apply boundary condition (2) first:

*Boundary condition* (2): $$\frac{du}{dr} = 0 + \frac{C_1}{0} = 0$$

Since $C_1/0$ is undefined (∞), the only way for $du/dr$ to equal zero at $r = 0$ is for $C_1$ to equal 0. An alternative way to think of this boundary condition is to say that $u$ must remain finite at the centerline of the pipe. Again this is possible only if constant $C_1$ is equal to 0.

$$C_1 = 0$$

Now we apply the first boundary condition,

*Boundary condition* (1):

$$u = \frac{-\rho g \sin\alpha}{4\mu} R^2 + 0 + C_2 = 0 \qquad \text{or} \qquad C_2 = \frac{\rho g \sin\alpha}{4\mu} R^2$$

Finally, Eq. 4 becomes

*Final result for axial velocity*: $$\boxed{u = \frac{\rho g \sin\alpha}{4\mu}\left(R^2 - r^2\right)} \quad (5)$$

The axial velocity profile is thus in the shape of a paraboloid, just as in Example 9-18.

**Step 6** Verify the results. You can plug in the velocity field to verify that all the differential equations and boundary conditions are satisfied.

The volume flow rate through the pipe is found by integrating Eq. 5 through the whole cross-sectional area of the pipe,

*Volume flow rate*:

$$\boxed{\dot{V} = \int_{\theta=0}^{2\pi}\int_{r=0}^{R} u\,dr = \frac{2\pi\rho g \sin\alpha}{4\mu}\int_{r=0}^{R}\left(R^2 - r^2\right) r\,dr = \frac{\pi R^4}{8\mu}\rho g \sin\alpha} \quad (6)$$

Since volume flow rate is also equal to the average axial velocity times cross-sectional area, we can easily determine the average axial velocity, $V$:

*Average axial velocity*: $$\boxed{V = \frac{\dot{V}}{A} = \frac{\frac{\pi R^4}{8\mu}\rho g \sin\alpha}{\pi R^2} = \frac{R^2}{8\mu}\rho g \sin\alpha} \quad (7)$$

***Discussion*** There is no such thing as an "inviscid" fluid. For example, if $\mu$ were zero in this problem, the axial velocity, volume flow rate, and average velocity would all go to infinity since $\mu$ appears in the denominator of Eqs. 5 through 7.

## Review Problems

### 9-101C

**Solution** We are to discuss the connection(s) between the incompressible flow approximation and the constant temperature approximation.

***Analysis*** For an incompressible flow, the density is assumed to be constant. In addition, the incompressible flow approximation usually implies that *all* fluid properties (viscosity, thermal conductivity, etc.) are constant as well. These assumptions go hand in hand because a flow with constant density implies a flow with little or no temperature changes and no buoyancy effects. Since viscosity is a strong function of temperature but generally a weak function of pressure, the fluid's viscosity is approximately constant whenever temperature is constant. When dealing with incompressible fluid flows, pressure variable $P$ is interpreted as the mechanical pressure $P_m$, and we don't need an equation of state. In effect, the equation of state is replaced by the assumption of constant density and constant temperature.

***Discussion*** Mechanical pressure $P_m$ is determined by the flow field, not by thermodynamics.

### 9-102C

**Solution** We are to name each equation, and then discuss its restrictions and its physical meaning.

***Analysis*** (*a*) This is the **continuity equation**. The form given here is valid for any fluid. It describes conservation of mass in a fluid flow.

(*b*) This is **Cauchy's equation**. The form given here is valid for any fluid. It describes conservation of linear momentum in a fluid flow.

(*c*) This is the **Navier-Stokes equation**. The form given here is valid for a specific type of fluid, namely an incompressible Newtonian fluid. The equation describes conservation of linear momentum in a fluid flow.

***Discussion*** It is important that you be able to recognize these notable equations of fluid mechanics.

### 9-103C

**Solution** We are to list the six steps used to solve fluid flow problem with the continuity and Navier-Stokes equations, for the case in which the fluid is incompressible and has constant properties.

***Analysis*** The steps are listed below:

**Step 1 Lay out the problem and the geometry. Identify all relevant dimensions and parameters.**
**Step 2 List all appropriate assumptions, approximations, simplifications, and boundary conditions.**
**Step 3 Write out and simplify the differential equations (continuity and the required components of Navier-Stokes) as much as possible.**
**Step 4 Solve (integrate) the differential equations. This leads to one or more constants of integration.**
**Step 5 Apply boundary conditions to obtain values for the constants of integration.**
**Step 6 Verify the results by checking that the flow field meets all the specifications and boundary conditions.**

***Discussion*** These steps are not always followed in the same order. For example, in CFD applications the boundary conditions are applied *before* the equations are integrated.

## 9-104C

**Solution**

(*a*) **True**: The unknowns for an incompressible flow problem with constant fluid properties are pressure and the three components of velocity. Density and viscosity are constants and are therefore not unknowns.

(*b*) **False**: The unknowns for a compressible flow problem are pressure, the three components of velocity, and the density. However, density is a thermodynamic function of pressure and temperature. Hence, temperature appears as an additional unknown, as does some kind of equation of state. In summary, there are actually at least 6 unknowns ($P$, $u$, $v$, $w$, $\rho$, and $T$). We therefore need 6 equations (continuity, 3 components of Navier-Stokes, equation of state, and energy). In addition, fluid properties such as viscosity may change as well, and we need either more equations or some kind of look-up table for these properties.

(*c*) **False**: Cauchy's equation contains additional unknowns – the components of the stress tensor, which must be written in terms of the velocity and pressure fields through some kind of constitutive equation.

(d) **True**: For an incompressible flow problem involving a Newtonian fluid, there are only 4 unknowns ($P$, $u$, $v$, and $w$). We therefore need only 4 equations (continuity and 3 components of Navier-Stokes).

## 9-105C

**Solution** We are to discuss the relationship between volumetric strain rate and the continuity equation.

***Analysis*** Volumetric strain rate is defined as the rate of increase of volume of a fluid element per unit volume. In a compressible flow field, the volume of a fluid particle may increase or decrease as it moves along in the flow, but its mass must remain constant. (This is a fundamental statement of conservation of mass of a system, since the fluid particle can be thought of as an infinitesimal system.) Mathematically it turns out that **volumetric strain rate is the sum of the three normal strain rates, and is identically zero for incompressible flow** (density cannot change, and hence volume cannot change). The continuity equation is based on the same fundamental principle of mass conservation. It is a differential form of the equation of conservation of mass. Its incompressible form also shows that the sum of the three normal strain rates must be zero. On the other hand, if the density is *not* constant, the sum of the three normal strain rates is not zero, but is still equal to the volumetric strain rate, which is also non-zero.

***Discussion*** Volumetric strain rate is derived and discussed in Chap. 4 as a kinematic property.

## 9-106

**Solution** For a given geometry and set of boundary conditions, we are to calculate the velocity and pressure fields, and plot the velocity profile.

***Assumptions*** The assumptions are identical to those of Example 9-17. We do not list them here.

***Analysis*** We obtain the velocity and pressure fields by following the step-by-step procedure for differential fluid flow solutions. Everything is identical to Example 9-17 except for the boundary condition at the wall. Boundary condition (1), the no-slip condition, becomes: at $x = 0$, $u = v = 0$. $w = V$. Steps 1 through 4 are otherwise identical, and the result is

*Result of integration of z momentum*: $$w = \frac{\rho g}{2\mu}x^2 + C_1 x + C_2 \qquad (1)$$

We continue, beginning with Step 5:

**Step 5** We apply boundary conditions (1) and (2) from Step 2 to obtain constants $C_1$ and $C_2$,

*Boundary condition* (1): $$w = 0 + 0 + C_2 = V \qquad C_2 = V$$

and

*Boundary condition* (2): $$\left.\frac{dw}{dx}\right)_{x=h} = \frac{\rho g}{\mu}h + C_1 = 0 \qquad C_1 = -\frac{\rho g h}{\mu}$$

Finally, Eq. 1 becomes

*Result*: $$\boxed{w = \frac{\rho g}{2\mu}x^2 - \frac{\rho g}{\mu}hx + V = \frac{\rho g x}{2\mu}(x - 2h) + V} \qquad (2)$$

Since $x < h$ in the film, the first term in Eq. 2 is negative, but the second term is positive. Depending on the relative magnitude of the terms, part or all of the vertical velocity may be positive. The pressure field is still $P = P_{atm}$ everywhere.

**Step 6** Verify the results. You can plug in the velocity field to verify that all the differential equations and boundary conditions are satisfied.

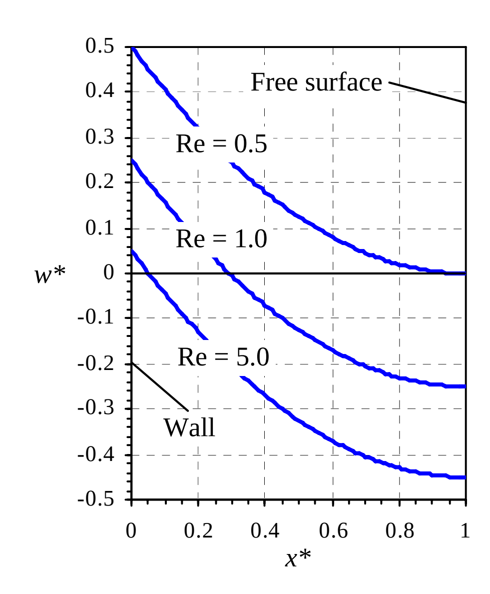



**FIGURE 1**
Velocity profiles for an oil film falling down a moving vertical wall. For all three Reynolds numbers, Fr = 0.5.

We nondimensionalize Eq. 2 by inspection: we let $x^* = x/h$ and $w^* = w\mu/(\rho g h^2)$. Eq. 2 becomes

*Nondimensional velocity profile*: $$w^* = \frac{x^*}{2}(x^* - 2) + \frac{V\mu}{\rho g h^2} \qquad (3)$$

We verify by inspection that when $V = 0$, Eq. 3 reduces to the velocity profile of Example 9-17. After some algebra we see that Eq. 3 can be re-written as

*Final nondimensional velocity profile*: $$w^* = \frac{x^*}{2}(x^* - 2) + \frac{\mathrm{Fr}^2}{\mathrm{Re}} \qquad (4)$$

where Froude number $\mathrm{Fr} = V/\sqrt{gh}$ and Reynolds number $\mathrm{Re} = \rho V h/\mu$. We plot the nondimensional velocity field in Fig. 1 for Fr = 0.5 and Re = 0.5, 1.0, and 5.0.

***Discussion*** Notice that the velocity profile has zero slope at the free surface regardless of the values of Fr and Re. For large enough $V$, the net mass flow rate is upward rather than downward.

## 9-107

**Solution** We are to calculate the volume flow rate per unit width of oil falling down a moving vertical wall, and then calculate the wall speed such that the net volume flow rate of oil is zero.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The wall is infinitely wide and very long so that all of the parallel flow, fully developed approximations of the previous problem hold.

***Analysis*** We calculate the volume flow rate per unit width by integration of the velocity:

*Volume flow rate per unit depth*:

$$\frac{\dot{V}}{L} = \int_0^h w\,dx = \int_0^h \left[\frac{\rho g x}{2\mu}(x-2h)+V\right]dx = \boxed{Vh - \frac{\rho g h^3}{3\mu}} \quad (1)$$

The volume flow rate is zero when the two terms in Eq. 1 cancel,

*Zero volume flow rate*:

$$\frac{\dot{V}}{L} = 0 \text{ when } Vh = \frac{\rho g h^3}{3\mu} \text{ or } \boxed{V = \frac{\rho g h^2}{3\mu}} \quad (2)$$

For an oil film of thickness 5.0 mm with $\rho$ = 888 kg/m$^3$ and $\mu$ = 0.80 kg/(m·s), we calculate $V$ using Eq. 2,

*Result for V*:

$$V = \frac{\rho g h^2}{3\mu} = \frac{(888 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(0.005 \text{ m})^2}{3(0.80 \text{ kg/m}\cdot\text{s})} = \mathbf{0.091\ m/s} \quad (3)$$

***Discussion*** For any $V$ greater than the value calculated in Eq. 3, the net oil flow is up, while for $V$ less than this value, the net oil flow is down. Since viscosity is in the denominator of Eq. 2, a low viscosity liquid (like water) would require a very large vertical velocity in order to achieve a net upward flow of the liquid.

---

## 9-108

**Solution** We are to define a $\psi$ that satisfies the continuity equation, and increases in the positive $z$ direction when the flow is from right to left in the $x$-$z$ plane.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the $x$-$z$ plane.

***Analysis*** We propose the following stream function,

*Stream function*:

$$\boxed{u = -\frac{\partial \psi}{\partial z} \qquad w = \frac{\partial \psi}{\partial x}} \quad (1)$$

We verify that the continuity equation is satisfied by Eq. 1,

*Steady, incompressible, 2-D continuity equation*:

$$\underbrace{\frac{\partial u}{\partial x}}_{-\frac{\partial^2 \psi}{\partial x \partial z}} + \underbrace{\frac{\partial w}{\partial z}}_{\frac{\partial^2 \psi}{\partial z \partial x}} = 0 \quad (2)$$

The only restriction is that $\psi$ must be a smooth function of $x$ and $z$. We check if we picked the proper signs by examining freestream flow from right to left in the $x$-$z$ plane:

*Freestream flow*:

$$u = -U \qquad w = 0 \qquad \psi = Uz + C \quad (3)$$

where $U$ is a positive constant and $C$ is an arbitrary constant. Thus we verify that as z increases, $\psi$ increases, and the flow is from right to left as desired.

***Discussion*** If we had defined $\psi$ with the opposite signs of Eq. 1, the flow would be from left to right as $\psi$ increases.

**9-109**

**Solution** We are to determine a relationship between constants *a*, *b*, *c*, *d*, and *e* that ensures incompressibility, and we are to determine the primary dimensions of each constant.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible (under certain restraints to be determined).

***Analysis*** We plug the velocity components into the incompressible continuity equation,

*Condition for incompressibility*:

$$\underbrace{\frac{\partial u}{\partial x}}_{az^2}+\underbrace{\frac{\partial v}{\partial y}}_{cxz}+\underbrace{\frac{\partial w}{\partial z}}_{3dz^2+2exz}=0 \qquad az^2+cxz+3dz^2+2exz=0 \qquad (1)$$

To guarantee incompressibility, the above equation must be satisfied everywhere. We equate similar terms to obtain the following relationships:

*Conditions for incompressibility*:

$$\boxed{a=-3d \qquad c=-2e} \qquad (2)$$

The units are found by observing that each component of the velocity field must be dimensionally homogeneous – each term must have dimensions of velocity. We examine each term:

$$\{axz^2\}=\{a\times L^3\}=\left\{\frac{L}{t}\right\} \qquad \boxed{\{a\}=\left\{\frac{1}{L^2t}\right\}}$$

$$\{by\}=\{b\times L\}=\left\{\frac{L}{t}\right\} \qquad \boxed{\{b\}=\left\{\frac{1}{t}\right\}}$$

$$\{cxyz\}=\{c\times L^3\}=\left\{\frac{L}{t}\right\} \qquad \boxed{\{c\}=\left\{\frac{1}{L^2t}\right\}}$$

$$\{dz^3\}=\{d\times L^3\}=\left\{\frac{L}{t}\right\} \qquad \boxed{\{d\}=\left\{\frac{1}{L^2t}\right\}}$$

$$\{exz^2\}=\{e\times L^3\}=\left\{\frac{L}{t}\right\} \qquad \boxed{\{e\}=\left\{\frac{1}{L^2t}\right\}}$$

***Discussion*** If Eq. 2 were not satisfied, the given velocity field might still represent a valid flow field, but density would have to vary with location in the flow field – in other words the flow would be *compressible*.

## 9-110

**Solution** We are to simplify the incompressible Navier-Stokes equation for the case of rigid body motion with arbitrary acceleration.

***Analysis*** We begin with the vector form of the incompressible Navier-Stokes equation,

*Incompressible Navier-Stokes equation:* $$\rho \frac{D\vec{V}}{Dt} = -\vec{\nabla}P + \rho\vec{g} + \mu\nabla^2\vec{V} \qquad (1)$$

In rigid body motion, $\vec{V}$ is not zero, but since the liquid moves as a solid body there is no *relative* motion between fluid particles. Thus the viscous term in Eq. 1 disappears. (Fluid particles do not rub against each other or shear against each other in any way, so the viscous term must vanish.) The material acceleration term $D\vec{V}/Dt$ is the acceleration following a fluid particle; hence it is identical to the imposed acceleration $\vec{a}$. Finally, $\vec{g} = -g\vec{k}$. Thus Eq. 1 reduces to

*Equation for rigid body acceleration:* $$\boxed{\vec{\nabla}P + \rho g\vec{k} = -\rho\vec{a}} \qquad (2)$$

***Discussion*** You can verify that Eq. 2 agrees with the rigid body acceleration equation of Chap. 3.

## 9-111

**Solution** We are to simplify the incompressible Navier-Stokes equation for the case of hydrostatics.

***Analysis*** We begin with the vector form of the incompressible Navier-Stokes equation,

*Incompressible Navier-Stokes equation:* $$\rho \frac{D\vec{V}}{Dt} = -\vec{\nabla}P + \rho\vec{g} + \mu\nabla^2\vec{V} \qquad (1)$$

In hydrostatics, $\vec{V}$ = 0 everywhere (no flow). Thus the first and last terms in Eq. 1 disappear. In addition, $\vec{g} = -g\vec{k}$. Thus Eq. 1 reduces to

*Hydrostatics equation:* $$\boxed{\vec{\nabla}P = -\rho g\vec{k}} \qquad (2)$$

***Discussion*** We verify from Eq. 2 that pressure does not change horizontally, but increases downward.

**9-112**

**Solution** We are to specify boundary conditions in terms of stream function.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible **3** The flow is two-dimensional.

***Analysis*** (*a*) For 2-D incompressible flow the difference in the value of the stream function between two streamlines is equal to the volume flow rate per unit width between the two streamlines. Since the entire flow is confined between the lower and upper channel walls, we know that stream function $\psi$ must be constant along the upper wall. We calculate $\psi$ on the upper channel wall as follows:

$$V_1 = \frac{\dot{V}}{H_1 W} = \frac{1}{H_1}\frac{\dot{V}}{W} = \frac{1}{H_1}\left(\psi_{\text{upper}} - \psi_{\text{lower}}\right) \qquad (1)$$

from which

$\psi_{\text{upper}}$: $$\psi_{\text{upper}} = \psi_{\text{lower}} + H_1 V = 0 + \left(0.12\text{ m}\right)\left(18.5\text{ m/s}\right) = \mathbf{2.22\ m^2/s} \qquad (2)$$

(*b*) Since the inlet flow is uniform, $\psi$ must increase linearly from $\psi_{\text{lower}}$ to $\psi_{\text{upper}}$ along the left edge of the computational domain. In equation form,

$\psi_{\text{left}}$: $$\psi_{\text{left}} = \psi_{\text{lower}} + \frac{\left(\psi_{\text{upper}} - \psi_{\text{lower}}\right)}{H_1} y = \frac{2.22\text{ m}^2\text{/s}}{0.12\text{ m}} y = \boxed{\left(18.5\text{ m/s}\right) y} \qquad (3)$$

We notice that Eq. 3 could have been obtained directly from $u = V_1 = \partial\psi/\partial y$.

(*c*) We have some options for the right edge of the computational domain. If that boundary is far enough away that it does not adversely affect the flow near the sudden contraction, we might specify a uniform velocity distribution along the right edge, similar to Eq. 3 above, but with a higher velocity determined by conservation of mass,

*Average outlet speed*: $$V_2 = V_1 \frac{H_1}{H_2} = \left(18.5\text{ m/s}\right)\frac{0.12\text{ m}}{0.046\text{ m}} = 48.26\text{ m/s}$$

In other words, we would specify

$\psi_{\text{right}}$: $$\boxed{\psi_{\text{right}} = \left(48.26\text{ m/s}\right) y} \qquad (4)$$

Eq. 4 is not a very good boundary condition because we know that viscous effects will surely slow down the flow near the walls – the velocity profile at the outlet will *not* be uniform.

A much better boundary condition (if the code permits it) is to specify that $\psi$ not change with $x$ along the right edge of the domain. Mathematically, we would specify

$\psi_{\text{right}}$: $$\frac{\partial \psi_{\text{right}}}{\partial x} = 0 \qquad (5)$$

You can see from the definition of $\psi$ that Eq. 5 is identical to forcing velocity component $v$ to be zero at the outlet. In other words, we specify that the flow at the outlet is *parallel*.

A third option would be to locate the right edge very far downstream so that the flow there is fully developed channel flow, for which we can specify the stream function as a function of $y$ along the edge. $\psi$ can be obtained from Problem 9-43.

***Discussion*** CFD and boundary conditions are discussed in detail in Chap. 15.

## 9-113

**Solution** For each equation we are to tell whether it is linear or nonlinear and explain.

***Analysis*** (*a*) The incompressible continuity equation is

*The incompressible continuity equation*:
$$\vec{\nabla}\cdot\vec{V}=0 \quad (1)$$

**This equation is linear**. There are no nonlinear terms.

(*b*) The compressible continuity equation is

*The compressible continuity equation*:
$$\frac{\partial\rho}{\partial t}+\vec{\nabla}\cdot\left(\rho\vec{V}\right)=0 \quad (2)$$

**This equation is nonlinear**. The second term has a product of two variables, $\rho$ and $\vec{V}$ – this is what makes the equation nonlinear.

(*c*) The incompressible Navier-Stokes equation is

*The incompressible Navier-Stokes equation*:
$$\rho\frac{D\vec{V}}{Dt}=-\vec{\nabla}P+\rho\vec{g}+\mu\nabla^2\vec{V} \quad (3)$$

**This equation is nonlinear**. The material acceleration term on the left can be written as

*The incompressible Navier-Stokes equation*:
$$\frac{D\vec{V}}{Dt}=\underbrace{\frac{\partial\vec{V}}{\partial t}}_{\text{Unsteady or local part}}+\underbrace{\left(\vec{V}\cdot\vec{\nabla}\right)\vec{V}}_{\text{Advective (or convective) part}} \quad (4)$$

The advective part of Eq. 4 contains products of variable $\vec{V}$ and derivatives of variable $\vec{V}$ – this is what makes the equation nonlinear.

***Discussion*** Density is treated as a constant in Eq. 3, and does not affect the nonlinearity of the equation. For *compressible* flow however, variable density causes the nonlinearity.

## 9-114

**Solution** We are to sketch some streamlines for boundary layer flow.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** We can offer only quantitative sketches of the streamlines. Since there is no flow reversal, we can be sure that **$\delta(x)$ is not a streamline**. In fact, streamlines must *cross* $\delta(x)$. Furthermore, at any given *y* location above the plate the fluid speed decreases as the boundary layer grows downstream. Hence, *the streamlines must diverge*. The bottom line is that the streamlines veer slightly upward away from the wall to compensate for the loss of speed in the boundary layer. Streamlines are sketched in Fig. 1.

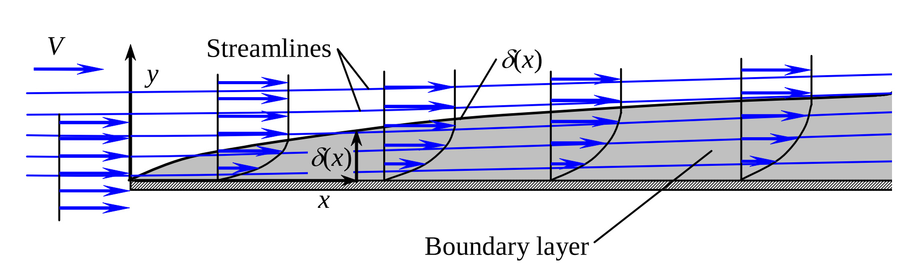



**FIGURE 1**
Streamlines above and within a flat plate boundary layer; since streamlines cross the curve $\delta(x)$, $\delta(x)$ cannot itself be a streamline of the flow. Furthermore, streamlines within the boundary layer veer up because of decreasing speeds within the boundary layer.

***Discussion*** As the boundary layer grows in thickness, more and more streamlines end up inside the boundary layer.

**9-115E**

**Solution** For a given axial velocity component in an axisymmetric flow field, we are to validate the incompressible approximation, generate the radial velocity component, generate an expression for the stream function, and then plot some streamlines and design the shape of the contraction.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is axisymmetric implying that $u_\theta = 0$ and there is no variation in the $\theta$ direction.

***Properties*** At room temperature and pressure, the speed of sound in air is about 1130 ft/s.

***Analysis*** (*a*) The maximum speed occurs in the test section, where the Mach number is

*Mach number*:
$$\text{Ma} = \frac{u_{z,L}}{c} = \frac{120\ \frac{\text{ft}}{\text{s}}}{1130\ \frac{\text{ft}}{\text{s}}} = 0.106 \qquad (1)$$

Since Ma is much less than 0.3, *the incompressible flow approximation is reasonable*.

(*b*) Between $z = 0$ and $z = L$, the axial velocity component is given by

*Axial velocity component*:
$$u_z = u_{z,0} + \frac{u_{z,L} - u_{z,0}}{L} z \qquad (2)$$

We use the incompressible continuity equation in cylindrical coordinates, simplified as follows for axisymmetric flow,

*Incompressible axisymmetric continuity equation*:
$$\frac{1}{r}\frac{\partial(ru_r)}{\partial r} + \frac{\partial(u_z)}{\partial z} = 0$$

After rearranging,

$$\frac{\partial(ru_r)}{\partial r} = -r\frac{\partial(u_z)}{\partial z} = -r\frac{u_{z,L} - u_{z,0}}{L} \qquad (3)$$

We integrate Eq. 3 with respect to *r*,

$$ru_r = -\frac{r^2}{2}\frac{u_{z,L} - u_{z,0}}{L} + f(z) \qquad (4)$$

Notice that since we performed a *partial* integration with respect to *r*, we add a function of the other variable *z* rather than simply a constant of integration. We divide all terms in Eq. 4 by *r* and recognize that the term with *f*(*z*) will go to infinity at the centerline of the contraction ($r = 0$) unless $f(z) = 0$. Our final expression for $u_r$ is thus

*Radial velocity component*:
$$\boxed{u_r = -\frac{r}{2}\frac{u_{z,L} - u_{z,0}}{L}} \qquad (5)$$

(*c*) The algebra for generating the stream function is identical to that of Problem 9-61 except for a change in notation. The result is thus

*Stream function*: $$\psi = \frac{r^2}{2}\left(u_{z,0} + \frac{u_{z,L} - u_{z,0}}{L}z\right) + \text{constant} \qquad (6)$$

The constant can be anything. We set it to zero for simplicity.

(*d*) First we calculate the axial speed at the entrance to the contraction. By conservation of mass,

$$u_{z,0}A_0 = u_{z,L}A_L \qquad \text{or} \qquad u_{z,0}\frac{\pi D_0^{\,2}}{4} = u_{z,L}\frac{\pi D_L^{\,2}}{4}$$

from which

$$u_{z,0} = u_{z,L}\frac{D_L^{\,2}}{D_0^{\,2}} = 120\frac{\text{ft}}{\text{s}} \times \frac{(1.5\text{ ft})^2}{(5.0\text{ ft})^2} = 10.8\frac{\text{ft}}{\text{s}}$$

We solve Eq. 6 for $r$ as a function of $z$ and plot several streamlines in Fig. 1,

*Streamlines*: $$r = \pm\sqrt{\frac{2\psi}{u_{z,0} + \frac{u_{z,L} - u_{z,0}}{L}z}} \qquad (7)$$

At the entrance of the contraction ($z$ = 0), the wall is at $r = D_0/2$ = 2.5 ft. Eq. 6 yields $\psi_{\text{wall}}$ = 33.75 ft$^3$/s for the streamline that passes through this point. This streamline thus represents the shape of the nozzle wall, and we have designed the nozzle shape.

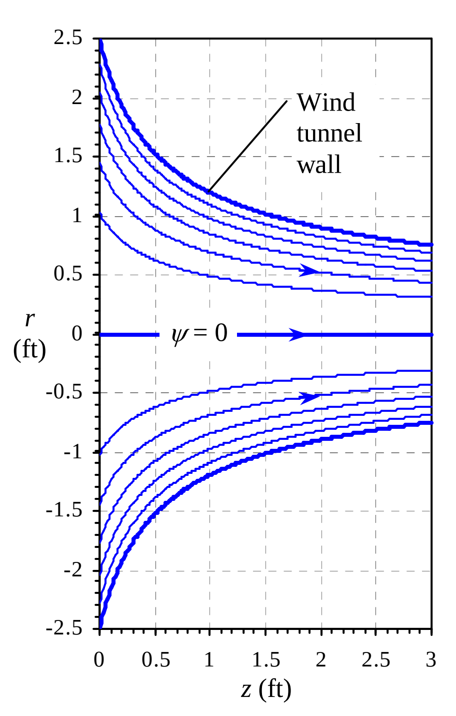



**FIGURE 1**
Streamlines for flow through an axisymmetric wind tunnel contraction.

***Discussion*** Since the boundary layers along the walls of the contraction are very small, the assumption about negligible friction effects is reasonable. This contraction shape should deliver the desired axial flow speed quite nicely.

## 9-116

**Solution** We are to determine the primary dimensions of $\psi$, nondimensionalize Eq. 1, and then plot several nondimensional streamlines for this flow field.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x-y* or *r-θ* plane.

***Analysis***

(*a*) There are several ways to calculate the primary dimensions of $\psi$. First, from Eq. 1 we see that

*Dimensions of stream function*:
$$\boxed{\{\psi\} = \left\{\frac{\dot{V}}{2\pi L}\right\} = \left\{\frac{\mathrm{L^3 t^{-1}}}{\mathrm{L}}\right\} = \left\{\frac{\mathrm{L^2}}{\mathrm{t}}\right\}} \tag{2}$$

We could also use the definition of $\psi$. Since velocities are obtained by spatial derivatives of $\psi$, $\psi$ must have an additional length dimension in the numerator compared to the dimensions of velocity. This reasoning also yields $\{\psi\} = \{\mathrm{L^2/t}\}$.

(*b*) The nondimensional form of the stream function is straightforward. Eq. 1 becomes

*Nondimensional stream function*:
$$\psi^* = -\arctan\frac{\sin 2\theta}{\cos 2\theta + \dfrac{1}{r^{*2}}} \tag{3}$$

(*c*) We solve Eq. 3 for $r^*$,

*Equation for nondimensional streamlines*:
$$\boxed{r^* = \pm\sqrt{\frac{\tan(-\psi^*)}{\sin 2\theta - \cos 2\theta \tan(-\psi^*)}}} \tag{4}$$

We pick the positive root to avoid negative radii. We plot several streamlines in the desired range in Fig. 1. The range of $\psi^*$ is 0 on the positive *x* axis to $-\pi$ on the positive *y* axis to $-2\pi$ on the negative *x* axis.

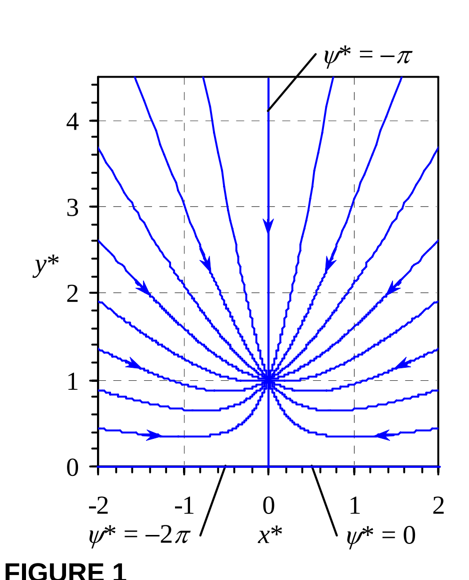



**FIGURE 1**
Nondimensional streamlines for flow into a vacuum cleaner attachment; $\psi^*$ is incremented uniformly from $2\pi$ (negative *x* axis) to 0 (positive *x* axis).

***Discussion*** The point ($x = 0$, $y = b$) is a singularity point with infinite velocity.

## 9-117

**Solution** We are to write Poisson's equation in standard form and discuss its similarities and differences compared to Laplace's equation.

***Analysis*** Poisson's equation in standard form is

*Poisson's equation*:
$$\boxed{\nabla^2\phi = s} \tag{1}$$

where $\phi$ is a dependent variable that is a function of space, $\nabla^2$ is the Laplacian operator, and *s* is the right hand side of the equation, which may be a function of space, but cannot be a function of $\phi$ itself. Poisson's equation is similar to Laplace's equation in that the left hand sides are identical. The difference is that Poisson's equation has a non-zero right hand side whereas the right hand side of Laplace's equation is zero. Note: Poisson's equation reduces to Laplace's equation if $s = 0$.

***Discussion*** We discuss Poisson's equation briefly in this chapter in relation to pressure correction algorithms used by CFD codes.

**9-118**

**Solution** We are to analyze this problem three ways: with the control volume technique, with the differential technique, and with dimensional analysis, and we are to compare the results.

***Assumptions*** **1** The flow is steady. **2** The flow is axisymmetric, incompressible, Newtonian, laminar, parallel, and fully developed ($u = u(r)$ only).

***Analysis*** (*a*) We use the head form of the energy equation from point 1 to point 2. Since there are no pumps, turbines, or minor losses the energy equation reduces to

*Energy equation*:

$$\frac{P_1}{\rho g}+\alpha_1\frac{V_1^2}{2g}+z_1=\frac{P_2}{\rho g}+\alpha_2\frac{V_2^2}{2g}+z_2+h_f \tag{1}$$

The pressure terms cancel since $P_1 = P_2 = P_{atm}$. The velocity terms cancel since the flow is fully developed. Upon substitution of the major head loss equation we have

*Reduced energy equation*:

$$\Delta z = z_1 - z_2 = h_f = f\frac{L}{D}\frac{V^2}{2g} \tag{2}$$

But for fully developed laminar pipe flow we know from Chap. 6 that the Darcy friction factor $f = 64/\text{Re}$. Thus Eq. 2 becomes

$$\Delta z = \frac{64}{\text{Re}}\frac{L}{D}\frac{V^2}{2g}=\frac{64\mu}{\rho VD}\frac{L}{D}\frac{V^2}{2g}=\frac{32\mu LV}{\rho D^2 g}$$

from which we can solve for average velocity $V$ through the pipe,

*V from control volume analysis*:

$$\boxed{V=\frac{\rho g D^2\Delta z}{32\mu L}} \tag{3}$$

(*b*) An exact analysis of this flow was performed in Problem 9-100. We refer to the solution of that problem and do not show the details here. The average velocity through the pipe was found to be

$$V=\frac{R^2}{8\mu}\rho g\sin\alpha$$

But $R = D/2$, and from the figure provided in the problem statement we see that $\sin\alpha = \Delta z/L$. Thus, our result is

*V from differential analysis*:

$$\boxed{V=\frac{\rho g D^2\Delta z}{32\mu L}} \tag{4}$$

The agreement with the result of Part (*a*) is exact.

(*c*) Finally we perform a dimensional analysis. We leave out the details, providing only a summary here; this is a good review of the material of Chap. 7. There are 7 parameters in the problem: $V$ as a function of $\rho$, $g$, $D$, $\Delta z$, $\mu$, and $L$. There are three primary dimensions represented in the problem, namely m, L, and t. Thus we expect 7-3 = 4 Πs. We choose three repeating variables, $\rho$, $g$, and $D$. The Πs are

*Dimensionless parameters*:

$$\Pi_1=\frac{V}{\sqrt{gD}}\qquad \Pi_2=\frac{\rho D\sqrt{gD}}{\mu}\qquad \Pi_3=\frac{\Delta z}{D}\qquad \Pi_4=\frac{L}{D}$$

The first Π is a Froude number and the second Π is a Reynolds number. The dimensionless relationship is

*Result of dimensional analysis*:

$$\boxed{\frac{V}{\sqrt{gD}}=f\left(\frac{\rho D\sqrt{gD}}{\mu},\frac{\Delta z}{D},\frac{L}{D}\right)} \tag{5}$$

To put the Πs of Eq. 5 into the form of Eq. 4 we do the following:

*Relationship between Πs*: $$\Pi_1 = \frac{\Pi_2\Pi_3}{32\Pi_4} \rightarrow \frac{V}{\sqrt{gD}} = \frac{\rho D\sqrt{gD}}{32\mu}\frac{\Delta z}{D}\frac{D}{L} \rightarrow \boxed{V = \frac{\rho g D^2 \Delta z}{32\mu L}} \qquad (6)$$

Thus we see that dimensional analysis is indeed consistent with the exact solution. Of course, we could not know the relationship of Eq. 6 by dimensional reasoning alone.

***Discussion*** The agreement between Parts (*a*), (*b*), and (*c*) is satisfying and emphasizes three different approaches to the same engineering problem.

---

## 9-119

**Solution** We are to analyze this problem two ways: with the exact (differential) technique, and with dimensional analysis, and we are to compare the results.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible, Newtonian, laminar, parallel, and fully developed ($u = u(y)$ only, where $x$ is in the direction of motion and $y$ is normal to the direction of motion). **3** We ignore aerodynamic drag on the block.

***Analysis*** (*a*) We draw a free-body diagram of the block in Fig. 1 and sum all the forces acting on it. There are only two forces in the $x$ direction: the x component of weight $W\sin\alpha$ and the force $\tau A$ due to viscous shear at the bottom surface of the block. Since the block slides at constant speed, these two forces must balance.

*Force balance*: $$W\sin\alpha = \tau A = \frac{\mu V A}{h} \qquad (1)$$

where we have used the exact analytical expression for the shear stress for Couette flow, namely $\tau = \mu(du/dy) = \mu V/h$. Solving for h,

*Exact solution for h*: $$\boxed{h = \frac{\mu V A}{W\sin\alpha}} \qquad (2)$$

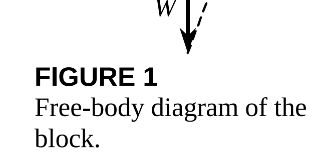



**FIGURE 1**
Free-body diagram of the block.

(*b*) We perform a dimensional analysis leaving out many of the details. There are 6 parameters in the problem: $h$ as a function of $V$, $A$, $W$, $\alpha$, and $\mu$. There are three primary dimensions represented in the problem, namely m, L, and t. Thus we expect 6-3 = 3 Πs. We choose three repeating variables, $V$, $A$, and $W$. The Πs are

*Dimensionless parameters*: $$\Pi_1 = \frac{h}{\sqrt{A}} \qquad \Pi_2 = \frac{\mu V\sqrt{A}}{W} \qquad \Pi_3 = \alpha$$

The dimensionless relationship is

*Result of dimensional analysis*: $$\boxed{\frac{h}{\sqrt{A}} = f\left(\frac{\mu V\sqrt{A}}{W}, \alpha\right)} \qquad (3)$$

To put the Πs of Eq. 3 into the form of Eq. 2 we do the following:

*Relationship between Πs*: $$\Pi_1 = \frac{\Pi_2}{\sin\Pi_3} \rightarrow \frac{h}{\sqrt{A}} = \frac{\mu V\sqrt{A}}{W\sin\alpha} \rightarrow \boxed{h = \frac{\mu V A}{W\sin\alpha}} \qquad (4)$$

Thus we see that dimensional analysis is indeed consistent with the exact solution. Of course, we could not know the relationship of Eq. 4 by dimensional reasoning alone.

***Discussion*** The agreement between Parts (*a*) and (*b*) is satisfying and emphasizes two different approaches to the same engineering problem.

---



# Solutions Manual for

# Fluid Mechanics: Fundamentals and Applications

# by Çengel & Cimbala

---

# CHAPTER 10

# APPROXIMATE SOLUTIONS OF THE NAVIER-STOKES EQUATION

---

## General and Introductory Problems, Modified Pressure, Fluid Statics

### 10-1C

**Solution** We are to discuss the difference between an "exact" solution and an approximate solution of the Navier-Stokes equation.

***Analysis*** **In an "exact" solution, we begin with the full Navier-Stokes equation**. As we solve the problem, some terms may drop out due to the specified geometry or other simplifying assumptions in the problem. **In an approximate solution, we eliminate some terms in the Navier-Stokes equation right from the start**. In other words, we *begin* with a reduced or simplified *approximate* form of the equation.

***Discussion*** The approximations are based on the class of flow problem and/or the *region* in which such approximations are appropriate (e.g. irrotational, boundary layer, etc.).

### 10-2C

**Solution** We are to label regions in a flow field where certain approximations are likely to be appropriate.

***Assumptions*** **1** The flow is incompressible. **2** The flow is steady in the mean (we ignore the unsteady flow field close to the rotating blades).

***Analysis*** A boundary layer grows along the floor, both upstream and downstream of the fan. The flow upstream of the fan is largely irrotational except very close to the floor. The air is nearly static far upstream and far above the fan. Downstream of the fan, the flow is most likely swirling and turbulent, and none of the approximations are expected to be appropriate there. In other words, the full Navier-Stokes equation must be solved in that region. **We sketch all these regions in Fig. 1**.

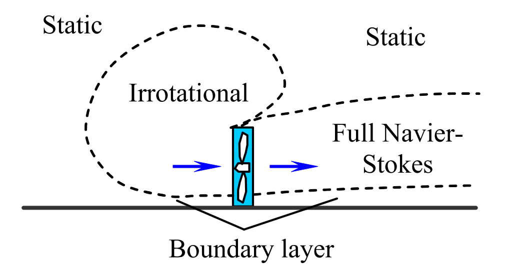



**FIGURE 1**
Regions of appropriate approximations for the flow produced by a box fan sitting on the floor of a large room.

***Discussion*** The regions sketched in Fig. 1 are not well defined, nor are they necessarily to scale.

### 10-3C

**Solution** We are to discuss the role of nondimensionalization of the Navier-Stokes equations.

***Analysis*** When we properly nondimensionalize the Navier-Stokes equation, **all the terms are re-written in the form of some nondimensional parameter times a quantity of order unity**. Thus, we can simply compare the orders of magnitude of the nondimensional parameters to see which terms (if any) can be ignored because they are very small compared to other terms. For example, if the Strouhal number is much smaller than the Euler number, we can ignore the term that contains the Strouhal number, but must retain the term that contains the Euler number.

***Discussion*** This method works only if the characteristic scales of the problem (length, speed, frequency, etc.) are chosen properly.

## 10-4C

**Solution** We are to discuss the most significant danger that arises with an approximate solution, and we are to come up with an example.

***Analysis*** The danger of an approximate solution of the Navier-Stokes equation is this: **If the approximation is not appropriate to begin with, our solution will be incorrect** – even if we perform all the mathematics correctly. There are many examples. For instance, we may assume that a boundary layer exists in a region of flow. However, if the Reynolds number is not large enough, the boundary layer is too thick and the boundary layer approximations break down. Another example is that we may assume a fluid statics region, when in reality there are swirling eddies in that region. The unsteady motion of the eddies makes the problem unsteady and dynamic – the approximation of fluid statics would be inappropriate.

***Discussion*** When you make an approximation and solve the problem, it is best to go back and verify that the approximation is appropriate.

---

## 10-5C

**Solution** We are to discuss the criteria used to determine whether an approximation of the Navier-Stokes equation is appropriate or not

***Analysis*** We determine if an approximation is appropriate **by comparing the orders of magnitude of the various terms in the equations of motion**. If the neglected terms are negligibly small compared to other terms, then the approximation is appropriate. If not, then it is not appropriate to neglect those terms.

***Discussion*** It is important that the proper scales be used for the nondimensionalization of the equation. Otherwise, the order of magnitude analysis may be incorrect.

---

## 10-6C

**Solution** We are to discuss the physical significance of the four nondimensional parameters in the nondimensionalized incompressible Navier-Stokes equation.

***Analysis*** The four parameters are discussed individually below:

- **Strouhal number**: St is the ratio of some characteristic flow time to some period of oscillation. If St << 1, the oscillation period is very large compared to the characteristic flow time, and the problem is quasi-steady; the unsteady term in the Navier-Stokes equation may be ignored. If St >> 1, the oscillation period is very short compared to the characteristic flow time, and the unsteadiness dominates the problem; the unsteady term must remain.
- **Euler number**: Eu is the ratio of a characteristic pressure difference to a characteristic pressure due to fluid inertia. If Eu << 1, pressure gradients are very small compared to inertial pressure, and the pressure term can be neglected in the Navier-Stokes equation. If Eu >> 1, the pressure term is very large compared to the inertial term, and must remain in the equation.
- **Froude number**: Fr is the ratio of inertial forces to gravitational forces. Note that Fr appears in the *denominator* of the nondimensionalized Navier-Stokes equation. If Fr << 1, gravitational forces are very large compared to inertial forces, and the gravity term must remain in the Navier-Stokes equation. If Fr >> 1, gravitational forces are negligible compared to inertial forces, and the gravity term in the Navier-Stokes equation can be ignored.
- **Reynolds number**: Re is the ratio of inertial forces to viscous forces. Note that Re appears in the *denominator* of the nondimensionalized Navier-Stokes equation. If Re << 1, viscous forces are very large compared to inertial forces, and the viscous term must remain. (In fact, it may dominate the other terms, as in creeping flow). If Re >> 1, viscous forces are negligible compared to inertial forces, and the viscous term in the Navier-Stokes equation can be ignored. Note that this applies only to regions outside of boundary layers, because the characteristic length scale for a boundary layer is generally much smaller than that for the overall flow.

***Discussion*** You must keep in mind that the approximations discussed here are appropriate only in certain *regions* of the flow field. In other regions of the same flow field, different approximations may apply.

---

**10-7C**

**Solution** We are to discuss the criterion for using modified pressure.

***Analysis*** **Modified pressure can be used only when there are *no free surface effects* in the problem**.

***Discussion*** Modified pressure is simply a combination of thermodynamic pressure and hydrostatic pressure. It turns out that if there are **no free surface effects**, the hydrostatic pressure component is independent of the flow pressure component, and these two can be separated.

---

**10-8C**

**Solution** We are to discuss which nondimensional parameter is eliminated by use of the modified pressure.

***Analysis*** Modified pressure effectively combines the effects of actual pressure and gravity. In the nondimensionalized Navier-Stokes equation in terms of modified pressure, **the Froude number disappears**. The reason Froude number is eliminated is because **the gravity term is eliminated from the equation.**

***Discussion*** Keep in mind that we can employ modified pressure only for flows without free surface effects.

---

## 10-9

**Solution** We are to plug the given scales for this flow problem into the nondimensionalized Navier-Stokes equation to show that only two terms remain in the region consisting of most of the tank.

***Assumptions*** **1** The flow is incompressible. **2** $d << D$. **3** $D$ is of the same order of magnitude as $H$.

***Analysis*** The characteristic frequency is taken as the inverse of the characteristic time, $f = 1/t_{\text{drain}}$. The Strouhal number is thus

*Strouhal number*:
$$\text{St} = \frac{fL}{V} = \frac{H}{t_{\text{drain}}V} \sim 1 \qquad (1)$$

St is of order of magnitude 1 since the order of magnitude of $t_{\text{drain}}$ is $H/V$. The Euler number is

*Euler number*:
$$\text{Eu} = \frac{P_0 - P_\infty}{\rho V^2} = \frac{\rho g H}{\rho V^2} \sim \frac{V_{\text{jet}}^{\ 2}}{V^2} \sim \frac{D^4}{d^4} \qquad (2)$$

where we have used the order of magnitude estimate that $V_{\text{jet}} \sim \sqrt{gH}$. We have also used conservation of mass, namely $V_{\text{jet}}d^2 = V_{\text{tank}}D^2$. Similarly, the Froude number is

*Froude number*:
$$\text{Fr} = \frac{V}{\sqrt{gH}} \sim \frac{V}{V_{\text{jet}}} \sim \frac{d^2}{D^2} \qquad (3)$$

Finally, the Reynolds number is

*Reynolds number*:
$$\text{Re} = \frac{\rho V H}{\mu} \sim \frac{\rho V D}{\mu} = \frac{\rho V_{\text{jet}} D}{\mu}\frac{V}{V_{\text{jet}}} \sim \text{Re}_{\text{jet}}\frac{d^2}{D^2} \qquad (4)$$

We plug Eqs. 1 through 4 into the nondimensionalized incompressible Navier-Stokes equation and compare orders of magnitude of each term,

*Nondimensionalized incompressible Navier-Stokes equation*:
$$\underbrace{[\text{St}]\frac{\partial \vec{V}^*}{\partial t^*}}_{\sim 1} + \underbrace{\left(\vec{V}^*\cdot\vec{\nabla}^*\right)\vec{V}^*}_{\sim 1} = \underbrace{-[\text{Eu}]\vec{\nabla}^* P^*}_{\sim \frac{D^2}{d^2}} + \underbrace{\left[\frac{1}{\text{Fr}^2}\right]\vec{g}^*}_{\sim \frac{D^2}{d^2}} + \underbrace{\left[\frac{1}{\text{Re}}\right]\vec{\nabla}^{*2}\vec{V}^*}_{\sim \text{Re}_{\text{jet}}\frac{d^2}{D^2}} \qquad (5)$$

Clearly, the first two terms (the unsteady and inertial terms) in Eq. 5 are negligible compared to the second two terms (the pressure and gravity terms) since $D >> d$. The last term (the viscous term) is a little trickier. We know that if the flow remains laminar, the order of magnitude of $\text{Re}_{\text{jet}}$ is at most $10^3$. Thus, in order for the viscous term to be of the same order of magnitude as the inertial term, $d^2/D^2$ must be of order of magnitude $10^{-3}$. Thus, provided that these criteria are met, the only two remaining terms in the Navier-Stokes equation are the pressure and gravity terms. The final dimensional form of the equation is the same as that of fluid statics,

*Incompressible Navier-Stokes equation for fluid statics:*
$$\vec{\nabla}P = \rho\vec{g} \qquad (6)$$

**The criteria for Carrie's approximation to be appropriate depends on the desired precision**. For 1% error, $D$ must be at least 10 times greater than $d$ to ignore the unsteady term and the inertial term. The viscous term, however, depends on the value of $\text{Re}_{\text{jet}}$. To be safe, Carrie should assume the highest possible value of $\text{Re}_{\text{jet}}$, for which we know from the above order of magnitude estimates that $D$ must be at least $10^{3/2}$ times greater than $d$.

***Discussion*** We cannot use the modified pressure in this problem since there is a free surface.

**10-10**

**Solution** We are to sketch the profile of modified pressure and shade in the region representing hydrostatic pressure.

***Assumptions*** **1** The flow is incompressible. **2** The flow is fully developed. **3** Gravity acts vertically downward. **4** There are no free surface effects in this flow field.

***Analysis*** By definition, modified pressure $P' = P + \rho gz$. So we add hydrostatic pressure component $\rho gz$ to the given profile for $P$ to obtain the profile for $P'$. Recall from Example 9-16, that for the case in which gravity does not act in the $x$-$z$ plane, the pressure would be constant along any slice $x = x_1$. Thus we infer that here (with gravity), the linear increase in $P$ as we move down vertically in the channel is due to hydrostatic pressure. Therefore, when we add $\rho gz$ to $P$ to obtain the modified pressure, it turns out that $P'$ is *constant* at this horizontal location.

**We show two solutions in Fig. 1**: (a) datum plane $z = 0$ located at the bottom wall, and (b) datum plane $z = 0$ located at the top wall. The shaded region in Fig. 1b represents the hydrostatic pressure component. $P'$ is constant along the slice $x = x_1$ for either case, and the datum plane can be drawn at any arbitrary elevation.

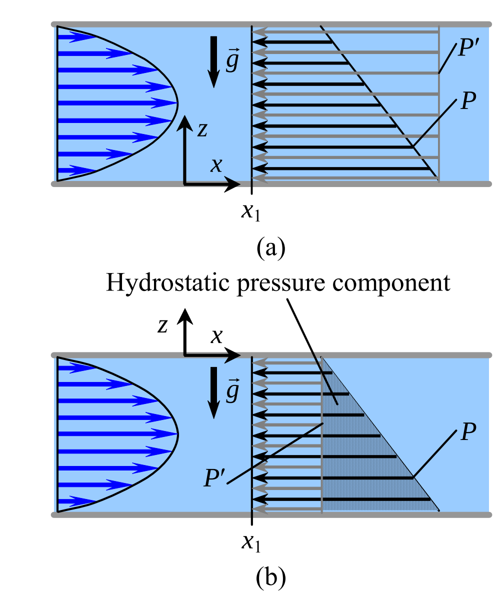



**FIGURE 1**
Actual pressure $P$ (black arrows) and modified pressure $P'$ (gray arrows) for fully developed planar Poiseuille flow. (a) Datum plane at bottom wall and (b) datum plane at top wall. The hydrostatic pressure component $\rho gz$ is the shaded area in (b).

***Discussion*** It should be apparent why it is advantageous to use modified pressure; namely, the gravity term is eliminated from the Navier-Stokes equation, and $P'$ is in general simpler than $P$.

---

**10-11**

**Solution** We are to discuss how modified pressure varies with downstream distance in planar Poiseuille flow.

***Assumptions*** **1** The flow is incompressible. **2** The flow is fully developed. **3** Gravity acts vertically downward. **4** There are no free surface effects in this flow field.

***Analysis*** For fully developed planar Poiseuille flow between two parallel plates, we know that pressure $P$ decreases linearly with $x$, the distance down the channel. Modified pressure is defined as $P' = P + \rho gz$. However, since the flow is horizontal, elevation $z$ does not change as we move axially down the channel. Thus we conclude that **modified pressure *P*′ decreases linearly with *x***. We sketch both $P$ and $P'$ in Fig. 1 at two axial locations, $x = x_1$ and $x = x_2$. The shaded region in Fig. 1 represents the hydrostatic pressure component $\rho gz$. Since channel height is constant, the hydrostatic component does not change with $x$. $P'$ is constant along any vertical slice, but its magnitude decreases linearly with $x$ as sketched.

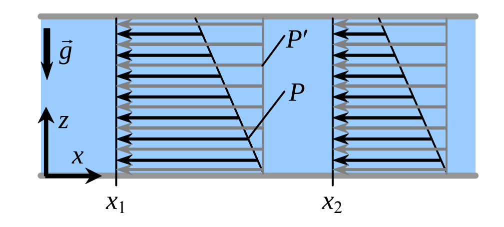



**FIGURE 1**
Actual pressure $P$ (black arrows) and modified pressure $P'$ (gray arrows) at two axial locations for fully developed planar Poiseuille flow.

***Discussion*** The pressure gradient $dP'/dx$ in terms of modified pressure is the same as the pressure gradient $\partial P/\partial x$ in terms of actual pressure.

---

**10-12**

**Solution** We are to generate an "exact" solution of the Navier-Stokes equation for fully developed Couette flow, using modified pressure. We are to compare to the solution of Chap. 9 that does *not* use modified pressure.

***Assumptions*** We number and list the assumptions for clarity:

**1** The plates are infinite in *x* and *z* (*z* is out of the page in the figure associated with this problem).
**2** The flow is steady.
**3** This is a parallel flow (we assume the *y* component of velocity, *v*, is zero).
**4** The fluid is incompressible and Newtonian, and the flow is laminar.
**5** Pressure *P* = constant with respect to *x*. In other words, there is no applied pressure gradient pushing the flow in the *x* direction; the flow establishes itself due to viscous stresses caused by the moving upper wall. In terms of modified pressure, $P'$ is also constant with respect to *x*.
**6** The velocity field is purely two-dimensional, which implies that $w = 0$ and $\frac{\partial}{\partial z}(\text{any velocity component}) = 0$.
**7** Gravity acts in the negative *z* direction.

***Analysis*** To obtain the velocity and pressure fields, we follow the step-by-step procedure outlined in Chap. 9.

**Step 1** Set up the problem and the geometry. See the figure associated with this problem.

**Step 2** List assumptions and boundary conditions. We have already listed seven assumptions. The boundary conditions come from imposing the no slip condition: (1) At the bottom plate ($y = 0$), $u = v = w = 0$. (2) At the top plate ($y = h$), $u = V$, $v = 0$, and $w = 0$. (3) At $z = 0$, $P = P_0$, and thus $P' = P + \rho g z = P_0$.

**Step 3** Write out and simplify the differential equations. We start with the continuity equation in Cartesian coordinates,

*Continuity*:
$$\frac{\partial u}{\partial x} + \underbrace{\frac{\partial v}{\partial y}}_{\text{Assumption 3}} + \underbrace{\frac{\partial w}{\partial z}}_{\text{Assumption 6}} = 0 \qquad \text{or} \qquad \frac{\partial u}{\partial x} = 0 \tag{1}$$

Equation 1 tells us that *u* is not a function of *x*. In other words, it doesn't matter where we place our origin – the flow is the same at any *x* location. I.e., the flow is fully developed. Furthermore, since *u* is not a function of time (Assumption 2) or *z* (Assumption 6), we conclude that *u* is at most a function of *y*,

*Result of continuity*:
$$u = u(y) \text{ only} \tag{2}$$

We now simplify the *x* momentum equation as far as possible:

$$\rho\left(\underbrace{\frac{\partial u}{\partial t}}_{\text{Assumption 2}} + \underbrace{u\frac{\partial u}{\partial x}}_{\text{Continuity}} + \underbrace{v\frac{\partial u}{\partial y}}_{\text{Assumption 3}} + \underbrace{w\frac{\partial u}{\partial z}}_{\text{Assumption 6}}\right) = \underbrace{-\frac{\partial P'}{\partial x}}_{\text{Assumption 5}} + \mu\left(\underbrace{\frac{\partial^2 u}{\partial x^2}}_{\text{Continuity}} + \frac{\partial^2 u}{\partial y^2} + \underbrace{\frac{\partial^2 u}{\partial z^2}}_{\text{Assumption 6}}\right) \quad \rightarrow \quad \frac{d^2 u}{dy^2} = 0 \tag{3}$$

All other terms in Eq. 3 have disappeared except for a lone viscous term, which must then itself equal zero. Notice that we have changed from a partial derivative ($\partial/\partial y$) to a total derivative (d/dy) in Eq. 3 as a direct result of Eq. 2. We do not show the details here, but you can show in similar fashion that every term except the pressure term in the *y* momentum equation goes to zero, forcing that lone term to also be zero,

*y momentum*:
$$\frac{\partial P'}{\partial y} = 0 \tag{4}$$

The same thing happens to the *z* momentum equation; the result is

*z momentum*:
$$\frac{\partial P'}{\partial z} = 0 \tag{5}$$

In other words, $P'$ is not a function of *y* or *z*. Since $P'$ is also not a function of time (Assumption 2) or *x* (Assumption 5), $P'$ is a constant,

*Result of y and z momentum*:
$$P' = \text{constant} = C_3 \tag{6}$$

**Step 4** Solve the differential equations. Continuity, *y* momentum, and *z* momentum have already been "solved", resulting in Eqs. 2 and 6. Equation 3 (*x* momentum) is integrated twice to get

*Integration of x momentum*: $$u = C_1 y + C_2 \qquad (7)$$

where $C_1$ and $C_2$ are constants of integration.

**Step 5** We apply boundary condition (3), $P' = P_0$ at $z = 0$. Eq. 6 yields $C_3 = P_0$, and

*Final solution for pressure field*: $$\boxed{P' = P_0 \quad \rightarrow \quad P = P_0 - \rho g z} \qquad (9)$$

We next apply boundary conditions (1) and (2) to obtain constants $C_1$ and $C_2$.

*Boundary condition* (1): $$u = C_1(0) + C_2 = 0 \quad \text{or} \quad C_2 = 0$$

and

*Boundary condition* (2): $$u = C_1(h) + 0 = V \quad \text{or} \quad C_1 = \frac{V}{h}$$

Finally, Eq. 7 becomes

*Final result for velocity field*: $$\boxed{u = V\frac{y}{h}} \qquad (10)$$

The velocity field reveals a simple linear velocity profile from $u = 0$ at the bottom plate to $u = V$ at the top plate.

**Step 6** Verify the results. You can plug in the velocity and pressure fields to verify that all the differential equations and boundary conditions are satisfied.

We verify that **the results are identical to those of Example 9-15**. Thus, we get the same result using modified pressure throughout the calculation as we do using the regular (thermodynamic) pressure throughout the calculation.

***Discussion*** Since there are no free surfaces in this problem, the gravity term in the Navier-Stokes equation is absorbed into the modified pressure, and the pressure and gravity terms are combined into one term. This is possible since flow pressure and hydrostatic pressure are uncoupled.

**10-13**

**Solution** We are to write all three components of the Navier-Stokes equation in terms of modified pressure, and show that they are equivalent to the equations with regular pressure. We are also to discuss the advantage of using modified pressure.

***Analysis*** In terms of modified pressure, the Navier-Stokes equation is written in Cartesian components as

*x component*:
$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial P'}{\partial x}+\mu\nabla^2 u \qquad (1)$$

and

*y component*:
$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial P'}{\partial y}+\mu\nabla^2 v \qquad (2)$$

and

*z component*:
$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial P'}{\partial z}+\mu\nabla^2 w \qquad (3)$$

The definition of modified pressure is

*Modified pressure*:
$$P' = P+\rho g z \qquad (4)$$

When Eq. 4 is plugged into Eqs. 1 and 2, the gravity term disappears since $z$ is independent of $x$ and $y$. The result is

*x component*:
$$\boxed{\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial P}{\partial x}+\mu\nabla^2 u} \qquad (5)$$

and

*y component*:
$$\boxed{\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial P}{\partial y}+\mu\nabla^2 v} \qquad (6)$$

However, when Eq. 4 is plugged into Eq. 3, the result is

*z component*:
$$\boxed{\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial P}{\partial z}-\rho g+\mu\nabla^2 w} \qquad (7)$$

Equations 5 through 7 are the appropriate components of the Navier-Stokes equation in terms of regular pressure, so long as gravity acts downward (in the $-z$ direction).

The advantage of using modified pressure is that the gravity term disappears from the Navier-Stokes equation.

***Discussion*** Modified pressure can be used only when there are no free surfaces.

## 10-14

**Solution** We are to sketch the profile of actual pressure and shade in the region representing hydrostatic pressure.

***Assumptions*** **1** The flow is incompressible. **2** Gravity acts vertically downward. **3** There are no free surface effects in this flow field.

***Analysis*** By definition, modified pressure $P' = P + \rho gz$. Thus, to obtain actual pressure $P$, we subtract the hydrostatic component $\rho gz$ from the given profile of $P'$. Using the given value of $P$ at the mid-way point as a guide, **we sketch the actual pressure in Fig. 1** such that the difference between $P'$ and $P$ increases linearly. In other words, we subtract the hydrostatic pressure component $\rho gz$ from the modified pressure $P'$ to obtain the profile for actual pressure $P$.

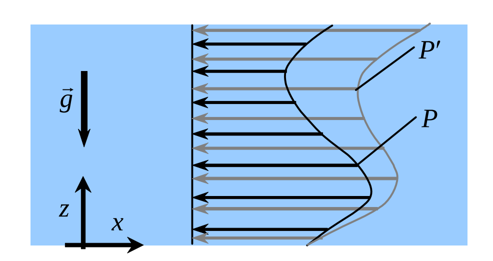



**FIGURE 1**
Actual pressure $P$ (black arrows) and modified pressure $P'$ (gray arrows) for the given pressure field.

***Discussion*** We assume that there are no free surface effects in the problem; otherwise modified pressure should not be used. The datum plane is set in the problem statement, but any arbitrary elevation could be used instead. If the datum plane were set at the top of the domain, $P'$ would be less than $P$ everywhere because of the negative values of $z$ in the transformation from $P$ to $P'$.

---

## 10-15

**Solution** We are to solve the Navier-Stokes equation in terms of modified pressure for the case of steady, fully developed, laminar flow in a round pipe. We are to obtain expressions for the pressure and velocity fields, and compare the actual pressure at the top of the pipe to that at the bottom of the pipe.

***Assumptions*** We make the same assumptions as in Example 9-18, except we use modified pressure $P'$ in place of actual pressure $P$.

***Analysis*** The Navier-Stokes equation with gravity, written in terms of modified pressure $P'$, is identical to the Navier-Stokes equation with no gravity, written in terms of actual pressure $P$. In other words, all of the algebra of Example 9-18 remains the same, except we use modified pressure $P'$ in place of actual pressure $P$. The velocity field does not change, and the result is

*Axial velocity field*:
$$\boxed{u = \frac{1}{4\mu}\frac{dP}{dx}\left(r^2 - R^2\right)} \qquad (1)$$

The modified pressure field is

*Modified pressure field*:
$$P' = P'(x) = P_1' + \frac{dP'}{dx}dx \qquad (2)$$

where $P_1'$ is the modified pressure at location $x = x_1$. In Example 9-18, the actual pressure varies only with $x$. In fact it decreases linearly with $x$ (note that the pressure gradient is negative for flow from left to right). Here, Eq. 2 shows that modified pressure behaves in the same way, namely $P'$ varies only with $x$, and in fact decreases linearly with $x$.

We simply subtract the hydrostatic pressure component $\rho gz$ from modified pressure $P'$ (Eq. 2) to obtain the final expression for actual pressure $P$,

*Actual pressure field*:
$$\boxed{P = P' - \rho gz \quad \rightarrow \quad P = P_1' + \frac{dP'}{dx}dx - \rho gz} \qquad (3)$$

Since the pipe is horizontal, the bottom of the pipe is lower than the top of the pipe. Thus, $z_{top}$ is greater than $z_{bottom}$, and therefore by Eq. 3 $P_{top}$ *is less than* $P_{bottom}$. This agrees with our experience that pressure increases downward.

***Discussion*** Since there are no free surfaces in this flow, the gravity term does not directly influence the velocity field, and a hydrostatic component is added to the pressure field. You can see the advantage of using modified pressure.

---

## Creeping Flow

### 10-16C

**Solution** We are to name each term in the Navier-Stokes equation, and then discuss which terms remain when the creeping flow approximation is made.

***Analysis*** The terms in the equation are identified as follows:

- **I** ***Unsteady*** **term**
- **II** ***Inertial*** **term**
- **III** ***Pressure*** **term**
- **IV** ***Gravity*** **term**
- **V** ***Viscous*** **term**

When the creeping flow approximation is made, **only terms III (pressure) and V (viscous) remain**. The other three terms are very small compared to these two and can be ignored. The significance is that all unsteady and inertial effects (terms I and II) have disappeared, as has gravity. We are left with a flow in which pressure forces and viscous forces must balance. Another significant result is that density has disappeared from the creeping flow equation, as discussed in the text.

***Discussion*** There are other acceptable one-word descriptions of some of the terms in the equation. For example, the inertial term can also be called the convective term, the advective term, or the acceleration term.

### 10-17

**Solution** We are to estimate the maximum speed of honey through a hole such that the Reynolds number remains below 0.1, at two different temperatures.

***Analysis*** The density of honey is equal to its specific gravity times the density of water,

*Density of honey*:

$$\rho_{\text{honey}} = SG_{\text{honey}}\rho_{\text{water}} = 1.42\left(998.0 \text{ kg/m}^3\right) = 1420 \text{ kg/m}^3 \qquad (1)$$

We convert the viscosity of honey from poise to standard SI units,

*Viscosity of honey at 20°*:

$$\mu_{\text{honey}} = 190 \text{ poise}\left(\frac{\text{g}}{\text{cm}\cdot\text{s}\cdot\text{poise}}\right)\left(\frac{\text{kg}}{1000 \text{ g}}\right)\left(\frac{100 \text{ cm}}{\text{m}}\right) = 19.0\frac{\text{kg}}{\text{m}\cdot\text{s}} \qquad (2)$$

Finally, we plug Eqs. 1 and 2 into the definition of Reynolds number, and set Re = 0.1 to solve for the maximum speed to ensure creeping flow,

*Maximum speed for creeping flow at 20°*:

$$V_{\max} = \frac{\text{Re}_{\max}\mu_{\text{honey}}}{\rho_{\text{honey}}D} = \frac{(0.1)(19.0 \text{ kg/m}\cdot\text{s})}{\left(1420 \text{ kg/m}^3\right)(0.0040 \text{ m})} = \mathbf{0.33\ m/s} \qquad (3)$$

At the higher temperature of 40°C, the calculations yield $V_{\max}$ = **0.035 m/s**. Thus, it is much easier to achieve creeping flow with honey at lower temperatures since the viscosity of honey increases rapidly as the temperature drops.

***Discussion*** We used Re < 0.1 as the maximum Reynolds number for creeping flow, but experiments reveal that in many flows, the creeping flow approximation is acceptable at Reynolds numbers as high as nearly 1.0.

**10-18**

**Solution** For each case we are to calculate the Reynolds number and determine if the creeping flow approximation is appropriate.

***Assumptions*** **1** The values given are characteristic scales of the motion.

***Properties*** For water at $T = 20^{o}$C, $\rho$ = 998.0 kg/m$^3$ and $\mu = 1.002 \times 10^{-3}$ kg/m·s. For unused engine oil at $T = 140^{o}$C, $\rho$ = 816.8 kg/m$^3$ and $\mu = 6.558 \times 10^{-3}$ kg/m·s. For air at $T = 30^{o}$C, $\rho$ = 1.164 kg/m$^3$ and $\mu = 1.872 \times 10^{-5}$ kg/m·s.

***Analysis*** (*a*) The Reynolds number of the microorganism is

$$\text{Re} = \frac{\rho D V}{\mu} = \frac{\left(998.0 \text{ kg/m}^3\right)\left(5.0\times10^{-6} \text{ m}\right)\left(0.2 \text{ mm/s}\right)}{1.002\times10^{-3} \text{ kg/m}\cdot\text{s}}\left(\frac{\text{m}}{1000 \text{ mm}}\right) = \mathbf{9.96\times10^{-4}}$$

Since Re << 1, the creeping flow approximation is certainly **appropriate**.

(*b*) The Reynolds number of the oil in the gap is

$$\text{Re} = \frac{\rho D V}{\mu} = \frac{\left(816.8 \text{ kg/m}^3\right)\left(0.0012 \text{ mm}\right)\left(20.0 \text{ m/s}\right)}{6.558\times10^{-3} \text{ kg/m}\cdot\text{s}}\left(\frac{\text{m}}{1000 \text{ mm}}\right) = \mathbf{2.99}$$

Since Re > 1, the creeping flow approximation is ***not*** **appropriate**.

(*c*) The Reynolds number of the fog droplet is

$$\text{Re} = \frac{\rho D V}{\mu} = \frac{\left(1.164 \text{ kg/m}^3\right)\left(10\times10^{-6} \text{ m}\right)\left(3.0 \text{ mm/s}\right)}{1.872\times10^{-5} \text{ kg/m}\cdot\text{s}}\left(\frac{\text{m}}{1000 \text{ mm}}\right) = \mathbf{1.87\times10^{-3}}$$

Since Re << 1, the creeping flow approximation is certainly **appropriate**.

***Discussion*** At room temperature, the oil viscosity increases by a factor of more than a hundred, and the Reynolds number of the bearing of Part (b) would be of order $10^{-2}$, which *is* in the creeping flow range.

## 10-19

**Solution** We are to estimate the speed and Reynolds number from a multiple-image photograph.

***Assumptions*** **1** The characteristic speed is taken as the average over 10 images.

***Properties*** For water at $T = 20^{o}$C, $\rho = 998.0$ kg/m$^3$ and $\mu = 1.002 \times 10^{-3}$ kg/m·s.

***Analysis*** By measurement with a ruler, we estimate the sperm's diameter as 2.4 μm, and it moves about 7.7 μm in 10 frames. This represents a time of

*Time for 10 frames*:
$$T = \frac{10 \text{ frames}}{200 \text{ frames/s}} = 0.050 \text{ s}$$

Thus the sperm's speed is

*Approximate speed*:
$$V = \frac{x}{T} = \frac{7.7 \ \mu\text{m}}{0.050 \text{ s}}\left(\frac{\text{m}}{10^6 \ \mu\text{m}}\right) = \mathbf{1.5\times10^{-4} \ m/s}$$

and its Reynolds number is

*Reynolds number*:
$$\text{Re} = \frac{\rho DV}{\mu} = \frac{\left(998.0 \text{ kg/m}^3\right)\left(2.4\times10^{-6} \text{ m}\right)\left(1.5\times10^{-4} \text{ m/s}\right)}{1.002\times10^{-3} \text{ kg/m}\cdot\text{s}} = 3.59\times10^{-4} \cong \mathbf{3.6\times10^{-4}}$$

Since Re << 1, the creeping flow approximation is certainly appropriate.

*Note*: Students' answers may differ widely since the measurements from the photograph are not very accurate. We report the final answer to only two significant digits because of the inherent error in measuring distances from the photograph.

***Discussion*** If you use the cell's length rather than its diameter as the characteristic length scale, Re increases by a factor of about two, but the flow is still well within the creeping flow regime.

## 10-20

**Solution** We are to compare the number of body lengths per second of a swimming human and a swimming sperm.

***Analysis*** We let *BLPS* denote "body lengths per second". For the human swimmer,

*Human*:
$$BLPS_{\text{human}} = \frac{100 \text{ m/min}}{1.8 \text{ m/body length}}\left(\frac{\text{min}}{60 \text{ s}}\right) = \mathbf{0.93 \ body \ length/s}$$

For the sperm, we use the speed calculated in Problem 10-19. The total body length of the sperm (head and tail) is about 40 μm, as measured from the figure.

*Sperm*:
$$BLPS_{\text{sperm}} = \frac{1.5\times10^{-4} \text{ m/s}}{40 \ \mu\text{m/body length}}\left(\frac{10^6 \ \mu\text{m}}{\text{m}}\right) = \mathbf{3.8 \ body \ length/s}$$

So, on an equal basis of comparison, the sperm swims faster than the human! This result is perhaps surprising since the human benefits from inertia, while the sperm feels no inertial effects. However, we must keep in mind that the sperm's body is designed to swim, while the human body is designed for multiple uses – it is not optimized for swimming.

*Note*: Students' answers may differ widely since the measurements from the photograph are not very accurate.

***Discussion*** Perhaps a more fair comparison would be between a *fish* and a sperm.

## 10-21

**Solution** We are to calculate how fast air must move vertically to keep a water drop suspended in the air.

***Assumptions*** **1** The drop is spherical. **2** The creeping flow approximation is appropriate.

***Properties*** For air at $T$ = 25°C, $\rho$ = 1.184 kg/m$^3$ and $\mu$ = 1.849 × 10$^{-5}$ kg/m·s. The density of the water at $T$ = 25°C is 997.0 kg/m$^3$.

***Analysis*** Since the drop is sitting still, its downward force must exactly balance its upward force when the vertical air speed $V$ is "just right". The downward force is the weight of the particle:

*Downward force on the particle*:
$$F_{\text{down}} = \pi \frac{D^3}{6} \rho_{\text{particle}} g \qquad (1)$$

The upward force is the aerodynamic drag force acting on the particle plus the buoyancy force on the particle. The aerodynamic drag force is obtained from the creeping flow drag on a sphere,

*Upward force on the particle*:
$$F_{\text{up}} = 3\pi\mu VD + \pi \frac{D^3}{6} \rho_{\text{air}} g \qquad (2)$$

We equate Eqs. 1 and 2, i.e., $F_{\text{down}} = F_{\text{up}}$,

*Balance*:
$$\pi \frac{D^3}{6} \left( \rho_{\text{particle}} - \rho_{\text{air}} \right) g = 3\pi\mu VD$$

and solve for the required air speed $V$,

$$V = \frac{D^2}{18\mu} \left( \rho_{\text{particle}} - \rho_{\text{air}} \right) g = \frac{\left(30\times10^{-6}\text{ m}\right)^2}{18\left(1.849\times10^{-5}\text{ kg/m}\cdot\text{s}\right)} \left[ \left(998.0 - 1.184\right)\text{ kg/m}^3 \right] \left(9.81\text{ m/s}^2\right) = \mathbf{0.0264\ m/s}$$

Finally, we must verify that the Reynolds number is small enough that the creeping flow approximation is appropriate.

*Check of Reynolds number*:
$$\text{Re} = \frac{\rho_{\text{air}} VD}{\mu} = \frac{\left(1.184\text{ kg/m}^3\right)\left(0.0264\text{ m/s}\right)\left(30\times10^{-6}\text{ m}\right)}{1.849\times10^{-5}\text{ kg/m}\cdot\text{s}} = \mathbf{0.0507}$$

Since Re << 1, The creeping flow approximation is appropriate.

***Discussion*** Notice that although air density does appear in the calculation of $V$, it is very small compared to the density of water. (If we ignore $\rho_{\text{air}}$ in that calculation, we get $V$ = 0.0265 m/s, an error of less than 0.4%. However, $\rho_{\text{air}}$ *is* required in the calculation of Reynolds number – to verify that the creeping flow approximation is appropriate.

---

## 10-22

**Solution** We are to discuss why density is not a factor in aerodynamic drag on a particle in creeping flow.

***Analysis*** It turns out that fluid density drops out of the creeping flow equations, since **the terms that contain $\rho$ in the Navier-Stokes equation are negligibly small compared to the pressure and viscous terms (which do not contain $\rho$)**. Another way to think about this is: In creeping flow, there is no fluid inertia, and since inertia is associated with fluid mass (density), density cannot contribute to the aerodynamic drag on a particle moving in creeping flow. In creeping flow, there is a balance between pressure forces and viscous forces, neither of which depend on fluid density.

***Discussion*** Density does have an *indirect* influence on creeping flow drag. Namely, $\rho$ is needed in the Reynolds number calculation, and Re determines whether the flow is in the creeping flow regime or not.

---

**10-23**

**Solution** We are to generate a characteristic pressure scale for flow through a slipper-pad bearing.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is two-dimensional in the *x*-*y* plane. **3** The creeping flow approximation is appropriate.

***Analysis*** The *x* component of the creeping flow momentum equation is

*x momentum*:
$$\frac{\partial P}{\partial x} \approx \mu \nabla^2 u \quad \rightarrow \quad \frac{\partial P}{\partial x} \approx \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \underbrace{\cancel{\frac{\partial^2 u}{\partial z^2}}}_{2-D} \right)$$

We plug in the characteristic scales to get

*Orders of magnitude*:
$$\underbrace{\frac{\partial P}{\partial x}}_{\frac{\Delta P}{L}} \approx \mu \underbrace{\frac{\partial^2 u}{\partial x^2}}_{\frac{V}{L^2}} + \mu \underbrace{\frac{\partial^2 u}{\partial y^2}}_{\frac{V}{h_0^2}} \tag{1}$$

The first term on the right of Eq. 1 is clearly much smaller than the second term on the right since $h_0 << L$. Equating the orders of magnitude of the two remaining terms,

*Characteristic pressure scale*:
$$\frac{\Delta P}{L} \sim \mu \frac{V}{h_0^2} \rightarrow \boxed{\Delta P \sim \frac{\mu V L}{h_0^2}} \tag{2}$$

***Discussion*** The characteristic pressure scale differs from that in the text because there are two length scales in this problem rather than just one.

**10-24**

**Solution** We are to find a characteristic velocity scale for $v$, compare the inertial terms of the $x$ momentum equation to the pressure and viscous terms, and discuss how the creeping flow equations can still be used even if Re is not small.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is two-dimensional in the $x$-$y$ plane. **3** Gravity forces are negligible.

***Analysis*** ($a$) We use the continuity equation to obtain the characteristic velocity scale for $v$,

*Continuity*:

$$\underbrace{\frac{\partial u}{\partial x}}_{\frac{V}{L}} + \underbrace{\frac{\partial v}{\partial y}}_{\frac{v}{h_0}} = 0 \rightarrow \boxed{v \sim \frac{Vh_0}{L}} \quad (1)$$

($b$) We analyze the orders of magnitude of each term in the steady, 2-D, incompressible $x$ momentum equation without gravity,

*x momentum*:

$$\underbrace{\rho u \frac{\partial u}{\partial x}}_{\frac{\rho V^2}{L}} + \underbrace{\rho v \frac{\partial u}{\partial y}}_{\rho \frac{Vh_0}{L}\frac{V}{h_0} = \frac{\rho V^2}{L}} = -\underbrace{\frac{\partial P}{\partial x}}_{\frac{\mu V}{h_0^2}} + \underbrace{\mu \frac{\partial^2 u}{\partial x^2}}_{\frac{\mu V}{L^2}} + \underbrace{\mu \frac{\partial^2 u}{\partial y^2}}_{\frac{\mu V}{h_0^2}} \quad (2)$$

where we have also used the result of Problem 10-23. The first viscous term of Eq. 2 is clearly much smaller than the second viscous term since $h_0 << L$. We multiply the order of magnitude of all the remaining terms by $L/(\rho V^2)$ to compare terms,

*Comparison of orders of magnitude*:

$$\underbrace{\rho u \frac{\partial u}{\partial x}}_{1} + \underbrace{\rho v \frac{\partial u}{\partial y}}_{1} = -\underbrace{\frac{\partial P}{\partial x}}_{\frac{\mu}{\rho V h_0}\frac{L}{h_0}} + \underbrace{\mu \frac{\partial^2 u}{\partial y^2}}_{\frac{\mu}{\rho V h_0}\frac{L}{h_0}} \quad (3)$$

We recognize the Reynolds number based on gap height, $\text{Re} = \rho V h_0/\mu$. Since the pressure and viscous terms contain the product of 1/Re, which is large for creeping flow, and $L/h_0$, which is also large, it is clear that **the inertial terms (left side of Eq. 3) are negligibly small compared to the pressure and viscous terms**.

($c$) Since the pressure and viscous terms contain the product of 1/Re and $L/h_0$, when $h_0 << L$, **the creeping flow equations can still be appropriate even if Reynolds number is not less than one**. For example, if $L/h_0 \sim 10{,}000$ and Re ~ 10, the pressure and viscous terms are still three orders of magnitude larger than the inertial terms.

***Discussion*** In the limit as $L/h_0 \rightarrow \infty$, the inertial terms disappear regardless of the Reynolds number. This limiting case is the Couette flow problem of Chap. 9.

## 10-25

**Solution** We are to analyze the $y$ momentum equation by order of magnitude analysis, and we are to comment about the pressure gradient $\partial P/\partial y$.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is two-dimensional in the $x$-$y$ plane. **3** Gravity forces are negligible.

***Analysis*** We analyze the orders of magnitude of each term in the steady, 2-D, incompressible $y$ momentum equation without gravity,

*y momentum*:

$$\underbrace{\rho u\frac{\partial v}{\partial x}}_{\rho V\frac{Vh_0}{L}\frac{1}{L}=\frac{\rho V^2 h_0}{L^2}} + \underbrace{\rho v\frac{\partial v}{\partial y}}_{\rho\frac{V^2 h_0^2}{L^2}\frac{1}{h_0}=\frac{\rho V^2 h_0}{L^2}} = -\underbrace{\frac{\partial P}{\partial y}}_{\frac{\mu VL}{h_0^3}} + \underbrace{\mu\frac{\partial^2 v}{\partial x^2}}_{\mu\frac{Vh_0}{L}\frac{1}{L^2}=\frac{\mu Vh_0}{L^3}} + \underbrace{\mu\frac{\partial^2 v}{\partial y^2}}_{\mu\frac{Vh_0}{L}\frac{1}{h_0^2}=\frac{\mu V}{Lh_0}} \quad (1)$$

The first viscous term of Eq. 1 is clearly much smaller than the second viscous term, since $h_0 << L$. We multiply the order of magnitude of all the remaining terms by $L^2/(\rho V^2 h_0)$ to compare terms,

*Comparison of orders of magnitude*:

$$\underbrace{\rho u\frac{\partial v}{\partial x}}_{1} + \underbrace{\rho v\frac{\partial v}{\partial y}}_{1} = -\underbrace{\frac{\partial P}{\partial y}}_{\frac{\mu}{\rho Vh_0}\left(\frac{L}{h_0}\right)^3} + \underbrace{\mu\frac{\partial^2 v}{\partial y^2}}_{\frac{\mu}{\rho Vh_0}\left(\frac{L}{h_0}\right)} \quad (2)$$

We recognize the Reynolds number based on gap height, Re = $\rho Vh_0/\mu$. Since the pressure and viscous terms contain the product of 1/Re, which is large for creeping flow, and $L/h_0$, which is also large, it is clear that the inertial terms (left side of Eq. 2) are negligibly small compared to the pressure and viscous terms. This is expected, of course, for creeping flow. Now we compare the pressure and viscous terms. Both contain 1/Re, but the pressure term has and additional factor of $(L/h_0)^2$, which is very large. Thus the pressure term is the only remaining term in Eq. 2. How can this be? Since there are no terms that can balance the pressure term, the pressure term itself must be very small. In other words, the $y$ momentum equation reduces to

*Final form of y momentum*:

$$\boxed{\frac{\partial P}{\partial y} \approx 0} \quad (3)$$

In other words, **pressure is a function of *x*, but a very weak, negligible function of *y***.

***Discussion*** The result here is very similar to that for boundary layers, where we also find that $\partial P/\partial y \approx 0$ through the boundary layer.

**10-26**

**Solution** We are to list boundary conditions and solve the *x* momentum equation for *u*. Then we are to nondimensionalize our result.

***Assumptions*** **1** The flow is steady and incompressible. **2** Gravity forces are negligible. **3** The flow is two-dimensional in the *x*-*y* plane. **4** *P* is not a function of *y*.

***Analysis*** (*a*) From the figure associated with this problem, we write two boundary conditions on *u*,

*Boundary condition* (1): $$\boxed{u = V \text{ at } y = 0 \text{ for all } x} \qquad (1)$$

and

*Boundary condition* (2): $$\boxed{u = 0 \text{ at } y = h \text{ for all } x} \qquad (2)$$

We note that *h* is not a constant, but rather a function of *x*.

(*b*) We write the creeping flow *x* momentum equation, and integrate once with respect to *y*, noting that *P* is not a function of *y*. This is a *partial* integration.

*Integration of x momentum*: $$\frac{\partial^2 u}{\partial y^2} = \frac{1}{\mu}\frac{dP}{dx} \quad \to \quad \frac{\partial u}{\partial y} = \frac{1}{\mu}\frac{dP}{dx}y + f_1(x)$$

We integrate again to obtain

*Second integration*: $$u = \frac{1}{2\mu}\frac{dP}{dx}y^2 + yf_1(x) + f_2(x) \qquad (3)$$

We apply boundary conditions to find the two unknown functions of *x*. From Eq. 1,

*Result of boundary condition* (1): $$f_2(x) = V$$

and from Eq. 2,

*Result of boundary condition* (2): $$f_1(x) = \frac{-V - \dfrac{1}{2\mu}\dfrac{dP}{dx}h^2}{h}$$

From these, the final expression for *u* is obtained,

*Final expression for u, dimensional*: $$\boxed{u(x,y) = V\left(1 - \frac{y}{h}\right) + \frac{h^2}{2\mu}\frac{dP}{dx}\frac{y}{h}\left(\frac{y}{h} - 1\right)} \qquad (4)$$

We recognize two distinct components of the velocity profile in Eq. 4, namely *a Couette flow component and a Poiseuille flow component*. Thus, the axial velocity is a superposition of Couette flow due to the moving bottom wall and Poiseuille flow due to the pressure gradient.

(*c*) We nondimensionalize Eq. 4 by applying the nondimensional variables given in the problem statement. After some algebra,

*Nondimensional expression for u*: $$\boxed{u^* = (1 - y^*) + \frac{h^{*2}}{2}\frac{dP^*}{dx^*}y^*(y^* - 1)} \qquad (5)$$

***Discussion*** Although we have a final expression for *u*, it is in terms of the pressure gradient *dP*/*dx*, which is not known. Pressure boundary conditions and further algebra are required to solve for the pressure field.

**10-27**

**Solution** We are to generate an expression for axial velocity for a slipper-pad bearing with arbitrary gap shape.

***Assumptions*** **1** The flow is steady and incompressible. **2** Gravity forces are negligible. **3** The flow is two-dimensional in the *x*-*y* plane. **4** *P* is not a function of *y*.

***Analysis*** In the solution of Problem 10-26, we never used the fact that *h*(*x*) was linear. In fact, our solution is in terms of *h*(*x*), the specific form of which was never specified. Thus, the solution of Problem 10-26 is still appropriate, and no further work needs to be done here. The result is

*Expression for u for arbitrary h(x)*:
$$u(x,y) = V\left(1-\frac{y}{h}\right)+\frac{h^2}{2\mu}\frac{dP}{dx}\frac{y}{h}\left(\frac{y}{h}-1\right) \qquad (1)$$

***Discussion*** As gap height *h*(*x*) changes, so does the pressure distribution.

---

**10-28**

**Solution** We are to prove the given equation for the slipper-pad bearing.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is two-dimensional in the *x*-*y* plane.

***Analysis*** We solve the 2-D continuity equation for *v* by integration,

*Continuity*:
$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \quad \rightarrow \quad \int_0^h \frac{\partial v}{\partial y}dy = -\int_0^h \frac{\partial u}{\partial x}dy \quad \rightarrow \quad v(h)-v(0) = -\int_0^h \frac{\partial u}{\partial x}dy \qquad (1)$$

But the no-slip condition tells us that *v* = 0 at both the bottom (*y* = 0) and top (*y* = *h*) plates. Thus Eq. 1 reduces to

*Result of continuity*:
$$\int_0^h \frac{\partial u}{\partial x}dy = 0 \qquad (2)$$

The 1-D Leibnitz theorem is discussed in Chap. 4 and is repeated here:

*1-D Leibnitz theorem*:
$$\frac{d}{dx}\int_{a(x)}^{b(x)} G(x,y)\,dy = \int_a^b \frac{\partial G}{\partial x}dy + \frac{db}{dx}G(x,b) - \frac{da}{dx}G(x,a) \qquad (3)$$

In our case (comparing Eqs. 2 and 3), *a* = 0, *b* = *h*(*x*), and *G* = *u*. Thus,

$$\frac{d}{dx}\int_0^h u\,dy = \int_0^h \frac{\partial u}{\partial x}dy + \frac{dh}{dx}u(x,h) \qquad (4)$$

But *u*(*h*) = 0 for all values of *x* (no-slip condition). Finally then, we combine Eqs. 2 and 4 to yield the desired result,

*Final result*:
$$\frac{d}{dx}\int_0^h u\,dy = 0$$

***Discussion*** This result could also be obtained by control volume conservation of mass. Now we finally have the means of calculating the pressure distribution in the slipper-pad bearing.

---

## 10-29

**Solution** We are to prove the given equation for flow through a 2-D slipper-pad bearing.

***Assumptions*** **1** The flow is steady and incompressible. **2** Gravity forces are negligible. **3** The flow is two-dimensional in the *x*-*y* plane. **4** *P* is not a function of *y*.

***Analysis*** We substitute the expression for *u* from Problem 10-26 into the equation of Problem 10-28,

$$\frac{d}{dx}\int_0^h u\,dy = 0 \quad \rightarrow \quad \frac{d}{dx}\int_0^h \left[V\left(1-\frac{y}{h}\right)+\frac{h^2}{2\mu}\frac{dP}{dx}\frac{y}{h}\left(\frac{y}{h}-1\right)\right]dy = 0 \qquad (1)$$

The integral in Eq. 1 is easily evaluated since both *h* and *dP*/*dx* are functions of *x* only. After some algebra,

$$\frac{d}{dx}\left[V\frac{h}{2}-\frac{h^3}{12\mu}\frac{dP}{dx}\right]=0 \qquad (2)$$

Finally, we take the *x* derivative, recognizing that *h* and *dP*/*dx* are functions of *x*,

*Steady, 2-D Reynolds equation for lubrication:*

$$\boxed{\frac{d}{dx}\left(h^3\frac{dP}{dx}\right)=6\mu V\frac{dh}{dx}} \qquad (3)$$

***Discussion*** For a given geometry (*h* as a known function of *x*), we can integrate Eq. 3 to obtain the pressure distribution along the slipper-pad bearing.

## 10-30

**Solution** We are to find the pressure distribution for flow through a 2-D slipper-pad bearing with linearly decreasing gap height and atmospheric pressure at both ends of the slipper-pad.

***Assumptions*** **1** The flow is steady and incompressible. **2** Gravity forces are negligible. **3** The flow is two-dimensional in the *x*-*y* plane. **4** *P* is not a function of *y*.

***Analysis*** We integrate the Reynolds equation of Problem 10-29, and rearrange:

*First integration*:

$$h^3\frac{dP}{dx}=6\mu Vh+C_1 \quad \rightarrow \quad \frac{dP}{dx}=6\mu Vh^{-2}+C_1h^{-3} \qquad (1)$$

where $C_1$ is a constant of integration. Next we substitute the given equation for *h*,

$$\frac{dP}{dx}=6\mu V\left(h_0+\alpha x\right)^{-2}+C_1\left(h_0+\alpha x\right)^{-3} \qquad (2)$$

Equation 2 is in the desired form, i.e., *dP*/*dx* as a function of *x*. We integrate Eq. 2,

*Second integration*:

$$P=-\frac{6\mu V}{\alpha}\left(h_0+\alpha x\right)^{-1}-\frac{C_1}{2\alpha}\left(h_0+\alpha x\right)^{-2}+C_2 \qquad (3)$$

where $C_2$ is a second constant of integration. We plug in the two boundary conditions on *P* to find constants $C_1$ and $C_2$, namely $P = P_{atm}$ at $x = 0$ and $P = P_{atm}$ at $x = L$. After some algebra, the results are

*Constants*:

$$C_1=-\frac{12\mu Vh_0h_L}{h_0+h_L} \quad \text{and} \quad C_2=P_{atm}+\frac{6\mu V}{\alpha\left(h_0+h_L\right)} \qquad (4)$$

with which we generate our final expression for *P* from Eq. 3. After some algebra,

*Pressure distribution*:

$$\boxed{P=P_{atm}+6\mu Vx\left[\frac{h_0-h_L+\alpha x}{\left(h_0+h_L\right)\left(h_0+\alpha x\right)^2}\right]} \qquad (5)$$

***Discussion*** There are other equivalent ways to write the expression for *P*, but Eq. 5 is about as compact as we can get.

**10-31E** *[Also solved using EES on enclosed DVD]*

**Solution** We are to calculate $\alpha$, we are to calculate $P_{gage}$ at a given $x$ location, and we are to plot nondimensional gage pressure as a function of nondimensional axial distance for the case of a slipper-pad bearing with linearly decreasing gap height. Finally, we are to estimate the total force that this slipper-pad bearing can support.

***Assumptions*** **1** The flow is steady and incompressible. **2** Gravity forces are negligible in the oil flow. **3** The flow is two-dimensional in the *x*-*y* plane. **4** *P* is not a function of *y*.

***Properties*** Unused engine oil at $T$ = 40°C: $\rho$ = 876.0 kg/m³, $\mu$ = 0.2177 kg/m·s.

***Analysis*** (*a*) The convergence is calculated by its definition (see Problem 10-30), and its tangent is also calculated,

$$\alpha = \frac{h_L - h_0}{L} = \frac{(0.0005 - 0.001)\text{ inch}}{1.0\text{ inch}} = \mathbf{-0.0005}$$

$$\rightarrow \qquad \tan\alpha = \mathbf{-0.0005}$$

Note that we must set $\alpha$ to radians when taking the tangent.

(*b*) At $x$ = 0.5 inches (0.0127 m), we calculate $P_{gage} = P - P_{atm}$ using the result of Problem 10-30; the gage pressure at the mid-way point is

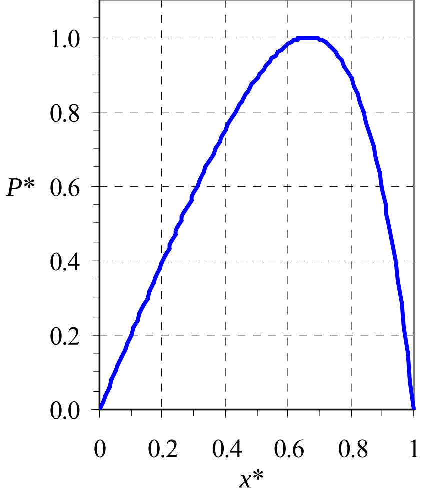



**FIGURE 1**
Nondimensional gage pressure in a slipper-pad bearing as a function of nondimensional axial distance along the slipper-pad.

$$P_{gage} = P - P_{atm} = 6\mu Vx\left[\frac{h_0 - h_L + \alpha x}{(h_0 + h_L)(h_0 + \alpha x)^2}\right]$$

$$= 6\left(0.2177\frac{\text{kg}}{\text{m}\cdot\text{s}}\right)\left(3.048\frac{\text{m}}{\text{s}}\right)(0.0127\text{ m})\left(\frac{\text{N s}^2}{\text{kg m}}\right)\left(\frac{\text{Pa m}^2}{\text{N}}\right)\left[\frac{\left[(2.54-1.27)\times10^{-5}\text{ m}\right] - 0.0005(0.0127\text{ m})}{\left[(2.54+1.27)\times10^{-5}\text{ m}\right]\left[2.54\times10^{-5}\text{ m} - 0.0005(0.0127\text{ m})\right]^2}\right]$$

$$= 2.32\times10^{7}\text{ Pa} = 3370\text{ psig} = \mathbf{229\ atm}$$

The gage pressure in the middle of the slipper-pad is more than 200 atmospheres. This is quite large, and illustrates how a small slipper-pad bearing can support a large amount of force.

(*c*) We repeat the calculations of Part (b) for values of $x$ between 0 and $L$. We nondimensionalize both $x$ and $P_{gage}$ using $x^* = x/L$ and $P^* = (P - P_{atm})h_0^2/\mu VL$. A plot of $P^*$ versus $x^*$ is shown in Fig. 1. The gage pressure is constrained to be zero at both ends of the pad, but reaches a peak near the middle, but more towards the end. For these conditions the maximum value of $P^*$ is 1.0.

(*d*) To calculate the total weight that the slipper-pad bearing can support, we integrate pressure over the surface area of the plate. We used the trapezoidal rule to integrate numerically in a spreadsheet. The result is

*Total vertical force (load)*: $$F_{load} = \int_{x=0}^{x=L} P_{gage}b\,dx = 62,600\text{ N} = \mathbf{14,100\ lbf}$$

You can also obtain a reasonable estimate by simply taking the average pressure in the gap times the area – this yields $F_{load}$ = 62,000 N = 13,900 lbf.

***Discussion*** This slipper-pad bearing can hold an enormous amount of weight (7 tons!) due to the extremely high pressures encountered in the oil passage.

**10-32**

**Solution** We are to discuss what happens to the load when the oil temperature increases.

***Analysis*** Oil viscosity appears only once in the equation for gap pressure. Thus, pressure and load increase linearly as oil viscosity increases. However, as the oil heats up, its viscosity goes down rapidly. For example, at $T = 40^{o}$C, $\mu = 0.2177$ kg/m·s, but at $T = 80^{o}$C, $\mu$ drops to 0.03232 kg/m·s. This is more than a factor of six decrease in viscosity for only a 20$^{o}$C increase in temperature. So, **the load would decrease rapidly as oil temperature rises**.

***Discussion*** This problem illustrates why engineers need to look at extreme operating conditions when designing products – just in case.

---

**10-33**

**Solution** We are to see if the Reynolds number is low enough that the flow can be approximated as creeping flow.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the *x-y* plane.

***Properties*** Unused engine oil at $T = 40^{o}$C: $\rho$ = 876.0 kg/m$^3$, $\mu = 0.2177$ kg/m·s.

***Analysis*** We base Re on the largest gap height, $h_0$,

*Reynolds number*:

$$\text{Re} = \frac{\rho h_0 V}{\mu} = \frac{\left(876.0\ \text{kg/m}^3\right)\left(2.54\times10^{-5}\ \text{m}\right)\left(3.048\ \text{m/s}\right)}{0.2177\ \text{kg/m}\cdot\text{s}} = 0.312$$

We see that the Reynolds number is less than one, but we cannot say that Re << 1. So, the flow is not really in the creeping flow regime. However, the creeping flow approximation is generally reasonable up to Reynolds numbers near one. Also, as discussed in Problem 10-24, **the creeping flow approximation is still reasonable** in this case since $L/h_0$ is so large.

***Discussion*** The error introduced by making the creeping flow approximation is probably less than the error associated with measurement of gap height.

---

**10-34** 

**Solution** We are to calculate how much the gap compresses when the load on the bearing is doubled.

***Analysis*** There are several ways to approach this problem: You can try to integrate the pressure distribution analytically to calculate the total load, or you can integrate numerically on a spreadsheet or math program. This is an "inverse" problem in that we can calculate the load for a given value of $h_0$, but we cannot do the reverse calculation directly – we must do it implicitly. One way to do this is graphically – plot load as a function of $h_0$, and pick off the value of $h_0$ where the load has doubled. Another way is by trial and error, or by a convergence technique like Newton's method. It turns out that **the load is doubled when $h_0$ = 0.0008535 inches (2.168 × 10$^{-5}$ m). This represents a decrease in initial gap height of about 14.7%.**

***Discussion*** The relationship between gap height and load is clearly nonlinear. When the load doubles, the gap height decreases by less than 15%.

---

## 10-35

**Solution** We are to estimate the speed at which a human being swimming in water would be in the creeping flow regime.

***Properties*** For water at $T = 20^{\circ}$C, $\rho = 998.0$ kg/m$^3$ and $\mu = 1.002 \times 10^{-3}$ kg/m·s.

***Analysis*** The characteristic length scale of a human body is of order 1 m. To be in the creeping flow regime, the Reynolds number of the body should be below 1. Thus,

$$\text{Re} = \frac{\rho L V}{\mu} \quad \rightarrow \quad V = \frac{\mu \text{Re}}{\rho \text{L}} = \frac{\left(1.002\times10^{-3}\ \text{kg/m}\cdot\text{s}\right)\left(1\right)}{\left(998\ \text{kg/m}^3\right)\left(1\ \text{m}\right)} \sim \mathbf{1\times10^{-6}\ m/s}$$

So, we would have to move at about one-millionth of a meter per second, or less. This speed is so slow that it is not measurable. Natural currents in the water, even in a "stagnant" pool of water, would be much greater than this. Hence, **we could never experience creeping flow in water**.

***Discussion*** If we were to use a Reynolds number of 0.1 instead of 1, the result would be even slower.

---

## Inviscid Flow

---

## 10-36C

**Solution** We are to discuss the approximation associated with the Euler equation.

***Analysis*** The Euler equation is simply the Navier-Stokes equation with *the viscous term neglected*; **it is therefore an inviscid approximation of the Navier-Stokes equation**. **The Euler equation is appropriate in high Reynolds number regions of the flow where net viscous forces are negligible, far away from walls and wakes.**

***Discussion*** The Euler equation is not appropriate very close to solid walls, since frictional forces are always present there. Note that the same Euler equation is appropriate in an *irrotational* region of flow as well.

---

## 10-37C

**Solution** We are to discuss the main difference between the steady, incompressible Bernoulli equation when applied to irrotational regions of flow vs. rotational but inviscid regions of flow.

***Analysis*** The Bernoulli equation itself is identical in these two cases, but the "constant" for the case of rotational but inviscid regions of flow is constant only along streamlines of the flow, not everywhere. **For irrotational regions of flow, the same Bernoulli constant holds everywhere**.

***Discussion*** A simple example is that of solid body rotation, which is rotational but inviscid. In this flow, as discussed in the text, the Bernoulli "constant" changes from one streamline to another.

---

**10-38**

**Solution** We are to show that the given vector identity is satisfied in Cartesian coordinates.

***Analysis*** We expand each term in the vector identity carefully. The first term is

$$\left(\vec{V}\cdot\vec{\nabla}\right)\vec{V} = \left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)\vec{i}+\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)\vec{j}+\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)\vec{k} \quad (1)$$

The second term is

$$\vec{\nabla}\left(\frac{V^2}{2}\right)=\frac{1}{2}\left[\left(\frac{\partial u^2}{\partial x}+\frac{\partial v^2}{\partial x}+\frac{\partial w^2}{\partial x}\right)\vec{i}+\left(\frac{\partial u^2}{\partial y}+\frac{\partial v^2}{\partial y}+\frac{\partial w^2}{\partial y}\right)\vec{j}+\left(\frac{\partial u^2}{\partial z}+\frac{\partial v^2}{\partial z}+\frac{\partial w^2}{\partial z}\right)\vec{k}\right]$$

which reduces to

$$\vec{\nabla}\left(\frac{V^2}{2}\right)=\left(u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial x}+w\frac{\partial w}{\partial x}\right)\vec{i}+\left(u\frac{\partial u}{\partial y}+v\frac{\partial v}{\partial y}+w\frac{\partial w}{\partial y}\right)\vec{j}+\left(u\frac{\partial u}{\partial z}+v\frac{\partial v}{\partial z}+w\frac{\partial w}{\partial z}\right)\vec{k} \quad (2)$$

The third term is

$$\vec{V}\times\left(\vec{\nabla}\times\vec{V}\right)=\left[v\left(\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y}\right)-w\left(\frac{\partial u}{\partial z}-\frac{\partial w}{\partial x}\right)\right]\vec{i}+\left[w\left(\frac{\partial w}{\partial y}-\frac{\partial v}{\partial z}\right)-u\left(\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y}\right)\right]\vec{j}+\left[u\left(\frac{\partial u}{\partial z}-\frac{\partial w}{\partial x}\right)-v\left(\frac{\partial w}{\partial y}-\frac{\partial v}{\partial z}\right)\right]\vec{k} \quad (3)$$

When we substitute Eqs. 1 through 3 into the given equation, we see that all the terms disappear, and **the equation is satisfied**. We show this for the *x* direction only (all terms with unit vector $\vec{i}$ ):

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial x}+w\frac{\partial w}{\partial x}-v\frac{\partial v}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}-w\frac{\partial w}{\partial x} \quad (4)$$

The algebra is similar for the $\vec{j}$ and $\vec{k}$ terms, and the vector identity is shown to be true for Cartesian coordinates.

***Discussion*** Since we have a vector identity, it must be true regardless of our choice of coordinate system.

**10-39**

**Solution** We are to use an alternative method to show that the Euler equation given in the problem statement reduces to the Bernoulli equation for regions of inviscid flow.

***Analysis*** We take the dot product of both sides of the equation with $\vec{V}$. The Euler equation dotted with velocity becomes

$$\vec{\nabla}\left(\frac{P}{\rho}+\frac{V^2}{2}+gz\right)\cdot\vec{V}=\left(\vec{V}\times\vec{\zeta}\right)\cdot\vec{V} \qquad (1)$$

The cross product on the right side of Eq. 1 is a vector that is perpendicular to $\vec{V}$. However, the dot product of two perpendicular vectors is zero by definition of the dot product. Thus, the right hand side of Eq. 1 is identically zero,

$$\vec{\nabla}\left(\frac{P}{\rho}+\frac{V^2}{2}+gz\right)\cdot\vec{V}=0 \qquad (2)$$

Now we use the same argument on the left hand side of Eq. 2, but in reverse. Namely, there are three ways for the dot product of the two vectors in Eq. 2 to be identically zero: (a) the first vector is zero,

*Option (a)*: $$\vec{\nabla}\left(\frac{P}{\rho}+\frac{V^2}{2}+gz\right)=0 \qquad (3)$$

(b) the second vector is zero,

*Option (b)*: $$\vec{V}=0 \qquad (4)$$

or (c) the two vectors are everywhere perpendicular to each other,

*Option (c)*: $$\vec{\nabla}\left(\frac{P}{\rho}+\frac{V^2}{2}+gz\right)\perp\vec{V} \qquad (5)$$

Option (a) represents the *restricted* case in which the quantity in parentheses in Eq. 3 is constant everywhere. Option (b) is the *trivial* case in which there is no flow (fluid statics). Option (c) is the most *general* option, and we work with Eq. 5. Since $\vec{V}$ is everywhere parallel to streamlines of the flow, $\vec{\nabla}\left(\frac{P}{\rho}+\frac{V^2}{2}+gz\right)$ must therefore be everywhere *perpendicular* to streamlines (Fig. 1). Finally, we argue that the gradient of a scalar is a vector that points perpendicular to an imaginary surface on which the scalar is constant. Thus, we argue that the scalar $\frac{P}{\rho}+\frac{V^2}{2}+gz$ must be *constant along a streamline*. Our final result is the steady incompressible Bernoulli equation for inviscid regions of flow,

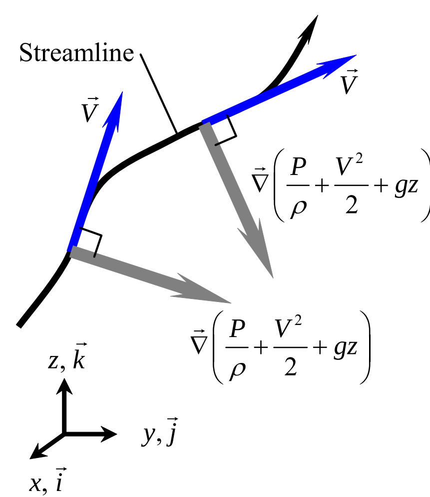



**FIGURE 1**

Along a streamline, $\vec{\nabla}\left(\frac{P}{\rho}+\frac{V^2}{2}+gz\right)$ is a vector everywhere perpendicular to the streamline; hence $\frac{P}{\rho}+\frac{V^2}{2}+gz$ is constant along the streamline.

$$\boxed{\frac{P}{\rho}+\frac{V^2}{2}+gz=\text{ constant along streamlines}} \qquad (6)$$

***Discussion*** Since we have a vector identity, it must be true regardless of our choice of coordinate system.

## 10-40

**Solution** We are to expand the Euler equation into Cartesian coordinates.

***Analysis*** We begin with the vector form of the Euler equation,

*Euler equation:*
$$\rho\left(\frac{\partial \vec{V}}{\partial t}+\left(\vec{V}\cdot\vec{\nabla}\right)\vec{V}\right)=-\vec{\nabla}P+\rho\vec{g} \quad (1)$$

The *x* component of Eq. 1 is

*x component*:
$$\boxed{\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial P}{\partial x}+\rho g_x} \quad (2)$$

The *y* component of Eq. 1 is

*y component*:
$$\boxed{\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial P}{\partial y}+\rho g_y} \quad (3)$$

The *z* component of Eq. 1 is

*z component*:
$$\boxed{\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial P}{\partial z}+\rho g_z} \quad (4)$$

***Discussion*** The expansion of the Euler equation into components is identical to that of the Navier-Stokes equation, except that the viscous terms are gone.

---

## 10-41

**Solution** We are to expand the Euler equation into cylindrical coordinates.

***Analysis*** We begin with the vector form of the Euler equation,

*Euler equation:*
$$\rho\left(\frac{\partial \vec{V}}{\partial t}+\left(\vec{V}\cdot\vec{\nabla}\right)\vec{V}\right)=-\vec{\nabla}P+\rho\vec{g} \quad (1)$$

We must be careful to include the "extra" terms in the convective acceleration. The *r* component of Eq. 1 is

*r component*:
$$\boxed{\rho\left(\frac{\partial u_r}{\partial t}+u_r\frac{\partial u_r}{\partial r}+\frac{u_\theta}{r}\frac{\partial u_r}{\partial \theta}-\frac{u_\theta^{\,2}}{r}+u_z\frac{\partial u_r}{\partial z}\right)=-\frac{\partial P}{\partial r}+\rho g_r} \quad (2)$$

The *y* component of Eq. 1 is

*θ component*:
$$\boxed{\rho\left(\frac{\partial u_\theta}{\partial t}+u_r\frac{\partial u_\theta}{\partial r}+\frac{u_\theta}{r}\frac{\partial u_\theta}{\partial \theta}+\frac{u_r u_\theta}{r}+u_z\frac{\partial u_\theta}{\partial z}\right)=-\frac{1}{r}\frac{\partial P}{\partial \theta}+\rho g_\theta} \quad (3)$$

The *z* component of Eq. 1 is

*z component*:
$$\boxed{\rho\left(\frac{\partial u_z}{\partial t}+u_r\frac{\partial u_z}{\partial r}+\frac{u_\theta}{r}\frac{\partial u_z}{\partial \theta}+u_z\frac{\partial u_z}{\partial z}\right)=-\frac{\partial P}{\partial z}+\rho g_z} \quad (4)$$

***Discussion*** The expansion of the Euler equation into components is identical to that of the Navier-Stokes equation, except that the viscous terms are gone.

---

**10-42**

**Solution** We are to calculate the pressure field and the shape of the free surface for solid body rotation of water in a container.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is rotationally symmetric, meaning that all derivatives with respect to $\theta$ are zero. **3** Gravity acts in the negative $z$ direction.

***Properties*** For water at $T = 20^{\circ}C$, $\rho = 998.0 \text{ kg/m}^3$ and $\mu = 1.002 \times 10^{-3}$ kg/m·s.

***Analysis*** We reduce the components of the Euler equation in cylindrical coordinates (Problem 10-41) as far as possible, noting that $u_r = u_z = 0$ and $u_\theta = \omega r$. The $\theta$ component disappears. The $r$ component reduces to

*r component of Euler equation*:
$$-\rho\frac{u_\theta^{\,2}}{r} = -\frac{\partial P}{\partial r} \quad \rightarrow \quad \frac{\partial P}{\partial r} = \rho\omega^2 r \qquad (1)$$

and the $z$ component reduces to

*z component of Euler equation*:
$$0 = -\frac{\partial P}{\partial z} - \rho g \quad \rightarrow \quad \frac{\partial P}{\partial z} = -\rho g \qquad (2)$$

We find $P(r,z)$ by cross integration. First we integrate Eq. 1 with respect to $r$,

$$P = \frac{\rho\omega^2 r^2}{2} + f(z) \qquad (3)$$

Note that we add a function of $z$ instead of a constant of integration since this is a *partial* integration. We take the $z$ derivative of Eq. 3, equate to Eq. 2, and integrate,

$$\frac{\partial P}{\partial z} = f'(z) = -\rho g \quad \rightarrow \quad f(z) = -\rho g z + C_1 \qquad (4)$$

Plugging Eq. 4 into Eq. 3 yields our expression for $P(r,z)$,

$$P = \frac{\rho\omega^2 r^2}{2} - \rho g z + C_1 \qquad (5)$$

Now we apply the boundary condition at the origin to find the value of constant $C_1$,

*Boundary condition*: At $r = 0$ and $z = 0$, $P = P_{atm} = C_1 \quad \rightarrow \quad C_1 = P_{atm}$

Finally, Eq. 5 becomes

*Pressure field*:
$$\boxed{P = \frac{\rho\omega^2 r^2}{2} - \rho g z + P_{atm}} \qquad (6)$$

At the free surface, we know that $P = P_{atm}$, and Eq. 6 yields the equation for the shape of the free surface,

*Free surface shape*:
$$\boxed{z_{surface} = \frac{\omega^2 r^2}{2g}} \qquad (7)$$

***Discussion*** Since we know the velocity field from the start, the Euler equation is not needed for obtaining the velocity field. Instead, it is used only to calculate the pressure field. Similarly, the continuity equation is identically satisfied and is not needed here.

---

**10-43**

**Solution** We are to calculate the pressure field and the shape of the free surface for solid body rotation of engine oil in a container.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is rotationally symmetric, meaning that all derivatives with respect to $\theta$ are zero. **3** Gravity acts in the negative $z$ direction.

***Analysis*** In Problem 10-42, water density appears only as a constant in the pressure equation. Thus, nothing is different here except the value of density, and **the results are identical to those of Problem 10-42**.

***Discussion*** In solid body rotation, the density of the fluid does not affect the shape of the free surface. For oil (less dense than water), pressure increases with depth at a slower rate compared to water.

---

**10-44**

**Solution** We are to calculate the Bernoulli constant for solid body rotation of water in a container.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is rotationally symmetric, meaning that all derivatives with respect to $\theta$ are zero. **3** Gravity acts in the negative $z$ direction.

***Analysis*** From Problem 10-42, we have the pressure field,

*Pressure field*:
$$P = \frac{\rho\omega^2 r^2}{2} - \rho g z + P_{\text{atm}} \qquad (1)$$

The Bernoulli equation for steady, incompressible, inviscid regions of flow is

$$\frac{P}{\rho} + \frac{V^2}{2} + gz = C_r = \text{constant along streamlines} \qquad (2)$$

The velocity field is $u_r = u_z = 0$ and $u_\theta = \omega r$, $V^2 = \omega^2 r^2$, and Eq. 2 becomes

$$C_r = \frac{P}{\rho} + \frac{\omega^2 r^2}{2} + gz \qquad (3)$$

Substitution of Eq. 1 into Eq. 3 yields the final expression for $C_r$,

*Bernoulli "constant"*:
$$C_r = \frac{\omega^2 r^2}{2} - gz + \frac{P_{\text{atm}}}{\rho} + \frac{\omega^2 r^2}{2} + gz \rightarrow \boxed{C_r = \frac{P_{\text{atm}}}{\rho} + \omega^2 r^2} \qquad (4)$$

***Discussion*** Streamlines in this flow field are circles about the $z$ axis (lines of constant $r$). The Bernoulli "constant" $C_r$ is constant along any given streamline, but changes from streamline to streamline. This is typical of rotating flow fields.

---

## 10-45

**Solution** For a given volume flow rate, we are to generate an expression for $u_r$ assuming inviscid flow, and then discuss the velocity profile shape for a real (viscous) flow.

***Assumptions*** **1** The flow remains radial at all times (no $u_\theta$ component). **2** The flow is steady, two-dimensional, and incompressible.

***Analysis*** If the flow were inviscid, we could not enforce the no-slip condition at the walls of the duct. At any $r$ location, the volume flow rate must be the same,

*Volume flow rate at any r location*: $$\dot{V} = u_r rb\Delta\theta \qquad (1)$$

where $\Delta\theta$ is the angle over which the contraction is bound (see Fig. 1). Thus,

$$u_r = \frac{\dot{V}}{rb\Delta\theta} \qquad (2)$$

At radius $r = R$, Eq. 2 becomes

*Radial velocity at r = R*: $$u_r(R) = \frac{\dot{V}}{Rb\Delta\theta} \qquad (3)$$

Upon substitution of Eq. 3 into Eq. 2, we get

*Radial velocity at any r location*: $$\boxed{u_r = \frac{R}{r}u_r(R)} \qquad (4)$$

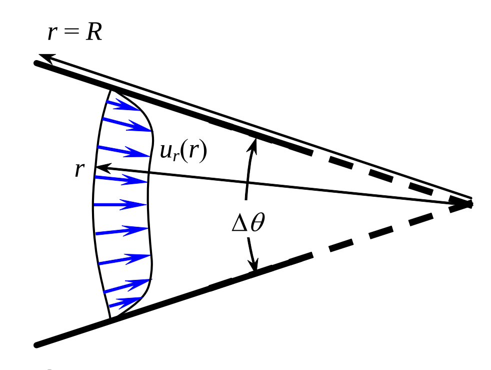



**FIGURE 1**
Possible shape of the velocity profile for a real (viscous) flow.

In other words, the radial velocity component increases as the reciprocal of $r$ as $r$ approaches zero (the origin).

In a real flow (with viscous effects), we would expect that the velocity near the center of the duct is somewhat larger, while that near the walls is somewhat smaller. Right at the walls, of course, the velocity is zero by the no-slip condition. In Fig. 1 is a sketch of what the velocity profile might look like in a real flow.

***Discussion*** In either case, the radial velocity is infinite at the origin. This is actually a portion of a line sink, as discussed in this chapter.

---

## 10-46

**Solution** We are to show that the region of flow given by this velocity field is inviscid.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the $x$-$y$ plane.

***Analysis*** We consider the viscous terms of the $x$ and $y$ momentum equations:

*x momentum viscous terms*: $$\mu\left(\underbrace{\frac{\partial^2 u}{\partial x^2}}_{0} + \underbrace{\frac{\partial^2 u}{\partial y^2}}_{0} + \underbrace{\frac{\partial^2 u}{\partial z^2}}_{0\text{ (2-D)}}\right) = 0 \qquad (1)$$

*y momentum viscous terms*: $$\mu\left(\underbrace{\frac{\partial^2 v}{\partial x^2}}_{0} + \underbrace{\frac{\partial^2 v}{\partial y^2}}_{0} + \underbrace{\frac{\partial^2 v}{\partial z^2}}_{0\text{ (2-D)}}\right) = 0 \qquad (2)$$

Since the viscous terms are identically zero in both components of the Navier-Stokes equation, **this region of flow can indeed be considered inviscid**.

***Discussion*** With the viscous terms removed, the Navier-Stokes equation is reduced to the Euler equation.

---

## Irrotational (Potential) Flow

### 10-47C

**Solution** We are to discuss the flow property that determines whether a region of flow is rotational or irrotational.

***Analysis*** The **vorticity** determines whether a region of flow is rotational or irrotational. Specifically, if the vorticity is zero (or negligibly small), the flow is approximated as irrotational, but if the vorticity is not negligibly small, the flow is rotational.

***Discussion*** Another acceptable answer is the rate of rotation vector or the angular velocity vector of a fluid particle.

### 10-48

**Solution** We are to show that the vorticity components are zero in an irrotational region of flow.

***Analysis*** The first component of vorticity becomes

*r-component of vorticity vector*:
$$\zeta_r = \frac{1}{r}\frac{\partial u_z}{\partial \theta} - \frac{\partial u_\theta}{\partial z} = \frac{1}{r}\frac{\partial^2 \phi}{\partial \theta \partial z} - \frac{1}{r}\frac{\partial^2 \phi}{\partial z \partial \theta} = 0$$

which is valid as long as $\phi$ is a smooth function of $\theta$ and $z$. Similarly, the second component of vorticity becomes

*$\theta$-component of vorticity vector*:
$$\zeta_\theta = \frac{\partial u_r}{\partial z} - \frac{\partial u_z}{\partial r} = \frac{\partial^2 \phi}{\partial z \partial r} - \frac{\partial^2 \phi}{\partial r \partial z} = 0$$

which is valid as long as $\phi$ is a smooth function of $r$ and $z$. Finally, the third component of vorticity becomes

*z-component of vorticity vector*:
$$\zeta_z = \frac{1}{r}\frac{\partial}{\partial r}\left(r u_\theta\right) - \frac{1}{r}\frac{\partial u_r}{\partial \theta} = \frac{1}{r}\frac{\partial^2 \phi}{\partial r \partial \theta} - \frac{1}{r}\frac{\partial^2 \phi}{\partial \theta \partial r} = 0$$

which is valid as long as $\phi$ is a smooth function of $r$ and $z$. Thus **all three components of vorticity are zero**.

***Discussion*** By mathematical identity, the velocity potential function is definable only when the vorticity vector is zero; therefore the results are not surprising. Note that in a three-dimensional flow, $\phi$ must be a smooth function of $r$, $\theta$, and $z$.

### 10-49

**Solution** We are to verify that the Laplace equation holds in an irrotational flow field in cylindrical coordinates.

***Analysis*** We plug in the components of velocity from Problem 10-48 into the left hand side of the Laplace equation in cylindrical coordinates,

*Laplace equation in cylindrical coordinates*:
$$\nabla^2 \phi = \frac{1}{r}\frac{\partial}{\partial r}\left(r u_r\right) + \frac{1}{r^2}\frac{\partial}{\partial \theta}\left(r u_\theta\right) + \frac{\partial u_z}{\partial z} \tag{1}$$

But since $r$ is not a function of $\theta$, we simplify Eq. 1 to

$$\nabla^2 \phi = \frac{1}{r}\frac{\partial}{\partial r}\left(r u_r\right) + \frac{1}{r}\frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z} \tag{2}$$

We recognize the terms on the right side of Eq. 2 as those of the incompressible form of the continuity equation in cylindrical coordinates; Eq. 2 is thus equal to zero, and the Laplace equation holds,

$$\boxed{\nabla^2 \phi = 0} \tag{3}$$

***Discussion*** The Laplace equation is valid for any incompressible irrotational region of flow, regardless of whether the flow is two- or three-dimensional.

## 10-50

**Solution** We are to identify regions in the flow field that are irrotational, and regions that are rotational.

***Assumptions*** **1** The air in the room would be calm if not for the presence of the hair dryer.

***Analysis*** **Flow in the air far away from the hair dryer and its jet is certainly irrotational. As the air approaches the inlet, it is irrotational except very close to the surface of the hair dryer. Flow in the jet is rotational, but flow outside of the jet can be approximated as irrotational**.

***Discussion*** Flow near solid walls is nearly always rotational because of the viscous rotational boundary layer that grows there. There are sharp velocity gradients in a jet, so the vorticity cannot be zero in that region, and the flow must be rotational in the jet as well.

---

## 10-51

**Solution** We are to compare the Bernoulli equation and its restrictions for inviscid, rotational regions of flow and viscous, irrotational regions of flow.

***Assumptions*** **1** The flow is incompressible and steady.

***Analysis*** The Bernoulli equation is the same in both cases, namely

*Steady incompressible Bernoulli equation*:
$$\boxed{\frac{P}{\rho}+\frac{V^2}{2}+gz=C} \qquad (1)$$

However, in an inviscid, rotational region of flow, Eq. 1 is applicable only along a streamline. The Bernoulli "constant" $C$ is constant along any particular streamline, but may change from streamline to streamline. In a viscous, irrotational region of flow, however, the Bernoulli constant is constant everywhere, even across streamlines. Thus, **the inviscid, rotational region of flow has more restrictions on the use of the Bernoulli equation**.

***Discussion*** In either case, the viscous terms in the Navier-Stokes equation disappear, but for different reasons.

---

## 10-52

**Solution** For a given set of streamlines, we are to sketch the corresponding set of equipotential curves and explain how we obtain them.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is two-dimensional in the plane of the figure associated with this problem. **3** The flow in the region shown in the figure is irrotational.

***Analysis*** **Some possible equipotential lines are sketched in Fig. 1**. We draw these based on the fact that the streamlines and equipotential curves must intersect at 90° angles. To find the "correct" shape, it helps to sketch a few extra streamlines in between the given ones to guide in construction of the equipotential curves. These "interpolated" streamlines are shown in Fig. 1 as thin, dotted blue lines.

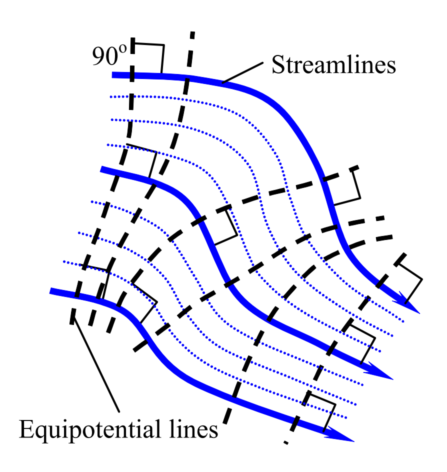



**FIGURE 1**
Possible equipotential curves (dashed black lines) and intermediate streamlines (dotted blue lines).

***Discussion*** The exact shape of the equipotential curves is not known, and individuals may sketch curves of other shapes that are equally valid. The important thing to emphasize is that the curves of constant $\phi$ are everywhere *perpendicular* to the streamlines.

---

## 10-53

**Solution** We are to discuss the role of the momentum equation in an irrotational region of flow.

***Assumptions*** **1** The flow is steady and incompressible. **2** The region of interest in the flow field is irrotational.

***Analysis*** Although it is true that the momentum equation is not required in order to solve for the velocity field, **it *is* required in order to solve for the *pressure* field**. In particular, the Navier-Stokes equation reduces to the Bernoulli equation in an irrotational region of flow.

***Discussion*** Mathematically, it turns out that in an irrotational flow field the continuity equation is *uncoupled* from the momentum equation, meaning that we can solve continuity for $\phi$ by itself, without need of the momentum equation. However, the momentum equation cannot be solved by itself.

---

## 10-54

**Solution** For a given velocity field, we are to assess whether the flow field is irrotational. If so, we are to generate an expression for the velocity potential function.

***Assumptions*** **1** The flow is steady. **2** The flow is incompressible. **3** The flow is two-dimensional in the *x-y* plane.

***Analysis*** For the flow to be irrotational, the vorticity must be zero. Since the flow is planar in the *x-y* plane, the only non-zero component of vorticity is in the *z* direction,

*z-component of vorticity*:
$$\zeta_z = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 - 0 = 0 \qquad (1)$$

Since the vorticity is zero, **this flow field can be considered irrotational**, and we should be able to generate a velocity potential function that describes the flow. In two dimensions we have

*Velocity components in terms of potential function*:
$$u = \frac{\partial \phi}{\partial x} \qquad v = \frac{\partial \phi}{\partial y} \qquad (2)$$

We pick one of these (the first one) and integrate to obtain an expression for $\phi$,

*Velocity potential function*:
$$\frac{\partial \phi}{\partial x} = u = ax + b \qquad \phi = a\frac{x^2}{2} + bx + f(y) \qquad (3)$$

Note that we have added a function of *y* rather than a constant of integration since we have performed a partial integration with respect to *x*. Using Eq. 2, we differentiate Eq. 3 with respect to *y* and equate the result to the *v* component of velocity,

$$\frac{\partial \phi}{\partial y} = f'(y) = v = -ay + c \qquad (4)$$

Equation 4 is integrated with respect to *y* to find function *f*(*y*),

$$f(y) = -a\frac{y^2}{2} + cy + C_1 \qquad (5)$$

This time, a constant of integration ($C_1$) is added since this is a total integration. Finally, we plug Eq. 5 into Eq. 3 to obtain our final expression for the velocity potential function,

*Result, velocity potential function*:
$$\boxed{\phi = a\frac{\left(x^2 - y^2\right)}{2} + bx + cy + C_1} \qquad (6)$$

***Discussion*** You should plug Eq. 6 into Eq. 2 to verify that it is correct.

---

**10-55**

**Solution** We are to discuss similarities and differences between two approximations: inviscid regions of flow and irrotational regions of flow.

***Assumptions*** **1** The flow is incompressible and steady.

***Analysis*** The two approximations are similar in that in both cases, the viscous terms in the Navier-Stokes equation drop out, leaving the Euler equation. Also, in both cases the Bernoulli equation results form integration of the Euler equation. However, these two approximations differ significantly from each other. **When making the inviscid flow approximation, we assume that the viscous terms are negligibly small**. A good example, as discussed in this chapter, is solid body rotation. In this case, although the fluid itself is viscous, all effects of viscosity are gone, and the flow field can be considered "inviscid" (although it is rotational). On the other hand, **the irrotational approximation is made when the vorticity (a measure of rotationality of fluid particles) is negligibly small**. In this case, viscosity still acts on fluid particles – it shears them and distorts them, yet the net rate of rotation of fluid particles is zero. In other words, in an irrotational region of flow, the net viscous force on a fluid particle is zero, but viscous stresses on the fluid particle are certainly *not* zero. Examples of irrotational, but viscous flows include any irrotational flow field with curved streamlines, such as a line vortex, a doublet, irrotational flow over a circular cylinder, etc. Freestream flow is both inviscid and irrotational since fluid particles do not shear or distort or rotate, and viscosity does not enter into the picture.

***Discussion*** In either case, the viscous terms in the Navier-Stokes equation disappear, but for different reasons. In the inviscid flow approximation, the viscous terms disappear because we neglect viscosity. In the irrotational flow approximation, the viscous terms disappear because they cancel each other out due to the fact that the vorticity (hence the rate of rotation) of fluid particles is negligibly small.

---

**10-56**

**Solution** We are to calculate the velocity components from a given potential function, verify that the velocity field is irrotational, and generate an expression for $\psi$.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is two-dimensional in the *x*-*y* plane. **3** The flow is irrotational in the region in which Eq. 1 applies.

***Analysis*** (*a*) The velocity components are found by taking the *x* and *y* partial derivatives of $\phi$,

*Velocity components*: $$\boxed{u = \frac{\partial \phi}{\partial x} = 10x + 2 \qquad v = \frac{\partial \phi}{\partial y} = -10y - 4} \tag{1}$$

(*b*) We plug in *u* and *v* from Eq. 1 into the *z* component of vorticity to get

*z-component of vorticity*: $$\zeta_z = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} = 0 - 0 = 0 \tag{2}$$

Since $\zeta_z = 0$, and the only component of vorticity in a 2-D flow in the *x*-*y* plane is in the *z* direction, the vorticity is zero, and **the flow is irrotational in the region of interest**.

(*c*) The stream function is found by integration of the velocity components. We begin by integrating the *x* component, $\partial\psi/\partial y = u$, and then taking the *x* derivative to compare with the known value of *v*,

$$\psi = 10xy + 2y + f(x) \quad \rightarrow \quad v = -\frac{\partial \psi}{\partial x} = -10y - f'(x) = -10y - 4 \tag{3}$$

From which we see that $f'(x) = 4$. Integrating with respect to *x*,

$$f(x) = 4x + \text{constant} \tag{4}$$

The constant is arbitrary since velocity components are always derivatives of $\psi$. Thus,

*Stream function*: $$\boxed{\psi = 10xy + 2y + 4x + \text{constant}} \tag{5}$$

***Discussion*** You can verify that the partial derivatives of Eq. 5 yield the same velocity components as those of Eq. 1.

---

## 10-57

**Solution** We are to show that the stream function for a planar irrotational region of flow satisfies the Laplace equation in cylindrical coordinates.

***Assumptions*** **1** This region of flow is planar in the $r$-$\theta$ plane. **2** The flow is incompressible. **3** This region of flow is irrotational.

***Analysis*** We defined stream function $\psi$ as

*Planar flow stream function in cylindrical coordinates:*

$$u_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} \qquad u_\theta = -\frac{\partial \psi}{\partial r} \tag{1}$$

We also know that for irrotational flow the vorticity must be zero. Since the only non-zero component of vorticity is in the $z$ direction,

*z-component of vorticity:*

$$\zeta_z = \frac{1}{r}\left(\frac{\partial(ru_\theta)}{\partial r} - \frac{\partial u_r}{\partial \theta}\right) = \frac{1}{r}\left(\frac{\partial}{\partial r}\left(-r\frac{\partial \psi}{\partial r}\right) - \frac{\partial}{\partial \theta}\left(\frac{1}{r}\frac{\partial \psi}{\partial \theta}\right)\right) = 0 \tag{2}$$

Since $r$ is not a function of $\theta$, it can come outside the derivative operator in the last term. Also, the negative sign in both terms can be disposed of. Thus,

*Result of irrotationality condition*:

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial \psi}{\partial r}\right) + \frac{1}{r^2}\frac{\partial^2 \psi}{\partial \theta^2} = 0 \tag{3}$$

Since Eq. 3 is the Laplace equation for 2-D planar flow in the $r$-$\theta$ plane, we have shown that **the stream function indeed satisfies the Laplace equation**.

***Discussion*** Since the Laplace equation for stream function is satisfied in Cartesian coordinates for the case of 2-D planar flow in the $x$-$y$ plane, it must also be satisfied for the same flow in cylindrical coordinates. All we have done is use a different coordinate system to describe the *same* flow.

---

## 10-58

**Solution** We are to write the Laplace equation in two dimensions ($r$ and $\theta$) in spherical polar coordinates.

***Assumptions*** **1** The flow is independent of angle $\phi$ (about the $x$ axis). **2** The flow is irrotational.

***Analysis*** We look up the Laplace equation in spherical polar coordinates in any vector analysis book. Ignoring derivatives with respect to $\phi$, we get

*Laplace equation, axisymmetric flow, (r,θ):*

$$\boxed{\frac{1}{r^2}\frac{\partial}{\partial r}\left(r^2\frac{\partial \phi}{\partial r}\right) + \frac{1}{r^2 \sin\theta}\frac{\partial}{\partial \theta}\left(\sin\theta\frac{\partial \phi}{\partial \theta}\right) = 0}$$

***Discussion*** Even though $\phi$ satisfies the Laplace equation in an irrotational region of flow, $\psi$ does *not* for the present case of axisymmetric flow.

---

**10-59**

**Solution** We are to prove that the given stream function exactly satisfies the continuity equation for the case of axisymmetric flow in spherical polar coordinates.

***Assumptions*** **1** The flow is axisymmetric, implying that there is no variation rotationally around the axis of symmetry. **2** The flow is incompressible.

***Analysis*** We plug the stream function into the continuity equation, and perform the algebra,

$$\frac{1}{r}\frac{\partial}{\partial r}\left(-\frac{1}{\sin\theta}\frac{\partial\psi}{\partial\theta}\right)+\frac{1}{\sin\theta}\frac{\partial}{\partial\theta}\left(\frac{1}{r}\frac{\partial\psi}{\partial r}\right)=0 \qquad (1)$$

since $\theta$ is not a function of $r$ and vice-versa, Eq. 1 can be rearranged as

$$-\frac{1}{r\sin\theta}\frac{\partial^2\psi}{\partial r\partial\theta}+\frac{1}{r\sin\theta}\frac{\partial^2\psi}{\partial\theta\partial r}=0 \qquad -\frac{\partial^2\psi}{\partial r\partial\theta}+\frac{\partial^2\psi}{\partial\theta\partial r}=0 \qquad (2)$$

**Equation 2 is identically satisfied as long as $\psi$ is a smooth function of $r$ and $\theta$.**

***Discussion*** If $\psi$ were *not* smooth, the order of differentiation ($r$ then $\theta$ versus $\theta$ then $r$) would be important and Eq. 2 would not necessarily be zero. In the definition of stream function, it is somewhat arbitrary whether we put the negative sign on $u_r$ or $u_\theta$, and you may find the opposite sign convention in other textbooks.

**10-60**

**Solution** We are to generate expressions for velocity potential function and stream function for the case of a uniform stream of magnitude $V$ inclined at angle $\alpha$.

***Assumptions*** **1** The flow is planar, incompressible, and irrotational. **2** The flow is uniform everywhere in the flow field, with magnitude $V$ and inclination angle $\alpha$.

***Analysis*** For planar flow in Cartesian coordinates, we write

$$u = \frac{\partial \phi}{\partial x} = \frac{\partial \psi}{\partial y} = V\cos\alpha \qquad v = \frac{\partial \phi}{\partial y} = -\frac{\partial \psi}{\partial x} = V\sin\alpha \qquad (1)$$

By integrating the first of these with respect to $x$, and then differentiating the result with respect to $y$, we generate an expression for the velocity potential function for a uniform stream,

$$\phi = Vx\cos\alpha + f(y) \qquad v = \frac{\partial \phi}{\partial y} = f'(y) = V\sin\alpha \qquad (2)$$

Integrating with respect to $y$,

$$f(y) = Vy\sin\alpha + \text{constant} \qquad (3)$$

The constant is arbitrary since velocity components are always derivatives of $\phi$. We set the constant to zero, knowing that we can always add an arbitrary constant later on if desired. Thus,

*Velocity potential function*: $$\boxed{\phi = Vx\cos\alpha + Vy\sin\alpha} \qquad (4)$$

We do a similar analysis for the stream function, beginning again with Eq. 1.

$$\psi = Vy\cos\alpha + g(x) \qquad v = -\frac{\partial \psi}{\partial x} = -g'(x) = V\sin\alpha \qquad (5)$$

Integrating with respect to $x$,

$$g(x) = -Vx\sin\alpha + \text{constant} \qquad (6)$$

The constant is arbitrary since velocity components are always derivatives of $\psi$. We set the constant to zero, knowing that we can always add an arbitrary constant later on if desired. Thus,

*Stream function*: $$\boxed{\psi = Vy\cos\alpha - Vx\sin\alpha} \qquad (7)$$

***Discussion*** You should be able to obtain the same answers by starting with the *opposite* equations in Eq. 1 (i.e., integrate first with respect to $y$ to obtain $\phi$ and with respect to $x$ to obtain $\psi$).

**10-61**

**Solution** We are to generate expressions for the stream function and the velocity potential function for a line source, beginning with the first equation above.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is irrotational in the region of interest. **3** The flow is two-dimensional in the *x*-*y* or $r$-$\theta$ plane.

***Analysis*** To find the stream function, we integrate the first equation with respect to $\theta$, and then differentiate with respect to the other variable *r*,

$$\frac{\partial\psi}{\partial\theta} = \frac{\dot{V}/L}{2\pi} \quad \rightarrow \quad \psi = \frac{\dot{V}/L}{2\pi}\theta + f(r) \quad \rightarrow \quad \frac{\partial\psi}{\partial r} = f'(r) = -u_\theta = 0 \qquad (1)$$

We integrate Eq. 1 to obtain

$$f(r) = \text{constant} \qquad (2)$$

We set the arbitrary constant of integration to zero since we can add back a constant as desired at any time without changing the flow. Thus,

*Line source at the origin*:
$$\boxed{\psi = \frac{\dot{V}/L}{2\pi}\theta} \qquad (3)$$

We perform a similar analysis for $\phi$ by beginning with the first equation:

$$\frac{\partial\phi}{\partial r} = \frac{\dot{V}/L}{2\pi r} \quad \rightarrow \quad \phi = \frac{\dot{V}/L}{2\pi}\ln r + f(\theta) \quad \rightarrow \quad \frac{\partial\phi}{\partial\theta} = f'(\theta) = ru_\theta = 0 \qquad (4)$$

We integrate Eq. 4 to obtain

$$f(\theta) = \text{constant}$$

We set the arbitrary constant of integration to zero since we can add back a constant as desired at any time without changing the flow. Thus,

*Line source at the origin*:
$$\boxed{\phi = \frac{\dot{V}/L}{2\pi}\ln r} \qquad (5)$$

***Discussion*** You can easily verify by differentiation that Eqs. 3 and 5 yield the correct velocity components. Also note that if $\dot{V}/L$ is negative, the flow field is that of a line sink rather than a line source.

**10-62**

**Solution** We are to generate expressions for the stream function and the velocity potential function for a line vortex.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is irrotational in the region of interest. **3** The flow is two-dimensional in the *x*-*y* or *r*-$\theta$ plane.

***Analysis*** To find the stream function we integrate the first equation with respect to $\theta$, and then differentiate with respect to the other variable *r*,

$$\frac{\partial\psi}{\partial\theta}=0 \quad \rightarrow \quad \psi=f(r) \quad \rightarrow \quad \frac{\partial\psi}{\partial r}=f'(r)=-u_\theta=-\frac{\Gamma}{2\pi r} \qquad (1)$$

We integrate Eq. 1 to obtain

$$f(r)=-\frac{\Gamma}{2\pi}\ln r+\text{constant} \qquad (2)$$

We set the arbitrary constant of integration to zero since we can add back a constant as desired at any time without changing the flow. Thus,

*Line vortex at the origin*:
$$\boxed{\psi=-\frac{\Gamma}{2\pi}\ln r} \qquad (3)$$

We perform a similar analysis for $\phi$. Beginning with the first equation:

$$\frac{\partial\phi}{\partial r}=0 \quad \rightarrow \quad \phi=f(\theta) \quad \rightarrow \quad \frac{\partial\phi}{\partial\theta}=f'(\theta)=ru_\theta=\frac{\Gamma}{2\pi} \qquad (4)$$

We integrate Eq. 4 to obtain

$$f(\theta)=\frac{\Gamma}{2\pi}\theta+\text{constant}$$

We set the arbitrary constant of integration to zero since we can add back a constant as desired at any time without changing the flow. Thus,

*Line vortex at the origin*:
$$\boxed{\phi=\frac{\Gamma}{2\pi}\theta} \qquad (5)$$

***Discussion*** You can easily verify by differentiation that Eqs. 3 and 5 yield the correct velocity components. Also note that if $\Gamma$ is positive, the vortex is counterclockwise, and if $\Gamma$ is negative, the vortex is clockwise.

## 10-63

**Solution** For a given stream function, we are to calculate the velocity potential function

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is two-dimensional in the $r$-$\theta$ plane. **3** The flow is approximated as irrotational.

***Analysis*** There are two ways to approach this problem: (1) Calculate the velocity components from the stream function, and then integrate to obtain $\phi$. (2) Superpose a freestream and a doublet to generate $\phi$ directly. We show both methods here.

**Method (1)**: We calculate the velocity components everywhere in the flow field by differentiating the stream function,

*Velocity components*:
$$u_r = \frac{1}{r}\frac{\partial \psi}{\partial \theta} = V_\infty \cos\theta\left(1-\frac{a^2}{r^2}\right) \qquad u_\theta = -\frac{\partial \psi}{\partial r} = -V_\infty \sin\theta\left(1+\frac{a^2}{r^2}\right) \tag{1}$$

Now we integrate to obtain the velocity potential function. We begin by integrating the expression for $u_r$ in Eq. 1,

$$u_r = \frac{\partial \phi}{\partial r} = V_\infty \cos\theta\left(1-\frac{a^2}{r^2}\right) \quad \rightarrow \quad \phi = V_\infty \cos\theta\left(r+\frac{a^2}{r}\right) + f(\theta) \tag{2}$$

We differentiate Eq. 2 with respect to $\theta$ and divide by $r$ to get

$$u_\theta = \frac{1}{r}\frac{\partial \phi}{\partial \theta} = -V_\infty \sin\theta\left(1+\frac{a^2}{r^2}\right) + \frac{f'(\theta)}{r} = -V_\infty \sin\theta\left(1+\frac{a^2}{r^2}\right) \tag{3}$$

Equation 3 reduces to $f'(\theta) = 0$, or $f(\theta)$ = constant. The constant is arbitrary, and we set it to zero for convenience. Hence, Eq. 2 reduces to

*Velocity potential, flow over a cylinder*:
$$\boxed{\phi = V_\infty \cos\theta\left(r+\frac{a^2}{r}\right)} \tag{4}$$

**Method (2)**: The velocity potential functions for a freestream and a doublet are superposed (added) to yield

*Superposition*:
$$\phi = V_\infty r\cos\theta + K\frac{\cos\theta}{r} \tag{5}$$

To find the doublet strength ($K$), we set the radial velocity component $u_r$ to zero at the cylinder surface ($r = a$),

$$u_r = \frac{\partial \phi}{\partial r} = V_\infty \cos\theta - K\frac{\cos\theta}{r^2} \quad \rightarrow \quad 0 = V_\infty \cos\theta - K\frac{\cos\theta}{a^2} \tag{6}$$

Equation 6 reduces to $K = a^2V_\infty$. Hence, Eq. 5 becomes

*Velocity potential, flow over a cylinder*:
$$\boxed{\phi = V_\infty \cos\theta\left(r+\frac{a^2}{r}\right)} \tag{7}$$

***Discussion*** Both methods yield the same answer, as they must.

---

## 10-64

**Solution** We are to discuss D'Alembert's paradox.

***Analysis*** D'Alembert's paradox states that **with the irrotational flow approximation, the aerodynamic drag force on any non-lifting body of any shape immersed in a uniform stream is zero**. It is a paradox because we know from experience that bodies in a flow field have non-zero aerodynamic drag.

***Discussion*** Irrotational flow over a non-lifting immersed body has neither pressure drag nor viscous drag. In a real flow, both of these drag components are present.

---

## Boundary Layers

### 10-65C

**Solution** We are to explain why the boundary layer approximation bridges the gap between the Euler equation and the Navier-Stokes equation.

***Analysis*** The Euler equation neglects the viscous terms compared to the inertial terms. For external flow around a body, this is a reasonable approximation over the majority of the flow field, except very close to the body, where viscous effects dominate. The Navier-Stokes equation, on the other hand, includes both viscous and inertial terms, but is much more difficult to solve. The boundary layer equations bridge the gap between these two: **we solve the simpler Euler equation away from walls, and then fit in a thin boundary layer to account for the no-slip condition at walls**.

***Discussion*** Students' discussions should be in their own words.

---

### 10-66C

**Solution**

(*a*) ***False***: If the Reynolds number at a given *x* location were to increase, all else being equal, viscous forces would decrease in magnitude relative to inertial forces, rendering the boundary layer thinner.

(*b*) ***False***: Actually, as *V* increases, so does Re, and the boundary layer thickness decreases with increasing Reynolds number.

(*c*) ***True***: Since $\mu$ appears in the denominator of the Reynolds number, Re decreases as $\mu$ increases, causing the boundary layer thickness to increase.

(d) ***False***: Since $\rho$ appears in the numerator of the Reynolds number, Re increases as $\rho$ increases, causing the boundary layer thickness to decrease.

---

### 10-67C

**Solution** We are to name three flows (other than flow along a wall) where the boundary layer approximation is appropriate, and we are to explain why.

***Analysis*** The boundary layer approximation is appropriate for the three basic types of shear layers: **wakes**, **jets**, and **mixing layers**. These flows have a predominant flow direction, and for high Reynolds numbers, the shear layer is very *thin*, causing the viscous terms to be much smaller than the inertial terms, just as in the case of a boundary layer along a wall.

***Discussion*** For the wake and the mixing layer, there is an irrotational outer flow in the streamwise direction, but for the jet, the flow outside the jet is nearly stagnant.

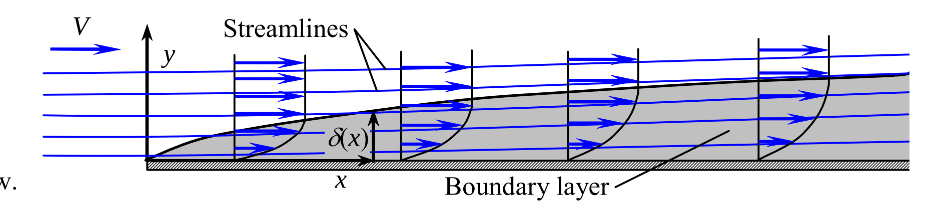
---

## 10-68C

**Solution** We are to sketch several streamlines and discuss whether the curve representing $\delta(x)$ is a streamline or not.

***Analysis*** **Five streamlines are sketched in Fig. 1**. In order to satisfy conservation of mass, the streamlines must cross the curve $\delta(x)$. Thus, $\delta(x)$ cannot itself be a streamline of the flow.

**FIGURE 1**
Several streamlines and the curve representing $\delta$ as a function of $x$ for a flat plate boundary layer. Since streamlines cross the curve $\delta(x)$, $\delta(x)$ cannot itself be a streamline of the flow.




***Discussion*** As the boundary layer grows in thickness, streamlines diverge slowly away from the wall (and become farther apart from each other) in order to conserve mass. However, the upward displacement of the streamlines is not as fast as the growth of $\delta(x)$.

## 10-69C

**Solution** We are to define trip wire and explain its purpose.

***Analysis*** A trip wire is **a rod or wire stretched normal to the streamwise direction along the wall**. Its purpose is to create a large disturbance in the laminar boundary layer that causes the boundary layer to "trip" to turbulence much more quickly than it would otherwise.

***Discussion*** Dimples on a golf ball serve the same purpose.

## 10-70

**Solution** We are to calculate the location of transition and turbulence along a flat plate boundary layer.

***Assumptions*** **1** The flow is incompressible and steady in the mean. **2** Freestream disturbances are small. **3** The surface of the plate is very smooth.

***Properties*** The density and viscosity of air at $T = 30^{\circ}$C are 1.164 kg/m$^3$ and $1.872 \times 10^{-5}$ kg/m·s respectively.

***Analysis*** Transition begins at the critical Reynolds number, which is approximately 100,000 for "clean" flow along a smooth flat plate. Thus,

*Beginning of transition*:

$$\mathrm{Re}_{x,\mathrm{critical}} = \frac{\rho V x_{\mathrm{critical}}}{\mu} = 100{,}000 \qquad (1)$$

Solving for $x$ yields

$$x_{\mathrm{critical}} = \frac{100{,}000\mu}{\rho V} = \frac{100{,}000\left(1.872\times10^{-5}\ \mathrm{kg/m\cdot s}\right)}{\left(1.164\ \mathrm{kg/m^3}\right)\left(25.0\ \mathrm{m/s}\right)} = 6.43\times10^{-2}\ \mathrm{m} \qquad (2)$$

That is, transition begins at $\boldsymbol{x \approx}$ **6 or 7 cm**. Fully turbulent flow in the boundary layer occurs at approximately 30 times $x_{\mathrm{critical}}$, at $\mathrm{Re}_{x,\mathrm{transition}} \approx 3 \times 10^6$. So, the boundary layer is expected to be fully turbulent by $\boldsymbol{x \approx}$ **2 m**.

***Discussion*** Final results are given to only one significant digit, since the locations of transition and turbulence are only approximations. The actual locations are influenced by many things, such as noise, roughness, vibrations, freestream disturbances, etc.

## 10-71E

**Solution** We are to assess whether the boundary layer on the surface of a fin is laminar or turbulent or transitional.

***Assumptions*** **1** The flow is steady and incompressible. **2** The fin surface is smooth.

***Properties*** The density and viscosity of water at $T$ = 40°F are 62.42 lbm/ft$^3$ and 1.038 × 10$^{-3}$ lbm/ft·s, respectively. The kinematic viscosity is thus $\nu$ = 1.663 × 10$^{-5}$ ft$^2$/s.

***Analysis*** Although the fin is not a flat plate, the flat plate boundary layer values are useful as a reasonable approximation to determine whether the boundary layer is laminar or turbulent. We calculate the Reynolds number at the trailing edge of the fin, using $c$ as the approximate streamwise distance along the flat plate,

$$\mathrm{Re}_x = \frac{Vc}{\nu} = \frac{(6.0 \text{ mi/hr})(1.6 \text{ ft})}{1.663\times10^{-5} \text{ ft}^2/\text{s}}\left(\frac{5280 \text{ ft}}{\text{mi}}\right)\left(\frac{\text{hr}}{3600 \text{ s}}\right) = 8.47\times10^5$$

The critical Reynolds number for transition to turbulence is 1 × 10$^5$ for the case of a smooth flat plate with very clean, low-noise freestream conditions. Our Reynolds number is higher than this. The engineering value of critical Reynolds number for real engineering flows is $\mathrm{Re}_{x,cr}$ = 5 × 10$^5$. Since $\mathrm{Re}_x$ is greater than $\mathrm{Re}_{x,cr}$, but less than $\mathrm{Re}_{x,transition}$ (30 × 10$^5$), **the boundary layer is most likely transitional, but may be fully turbulent by the trailing edge of the fin**.

***Discussion*** In a real-life situation, the freestream flow is not very "clean" – there are eddies and other disturbances, the fin surface is not perfectly smooth, and the vehicle may be vibrating. Thus, transition and turbulence are likely to occur much earlier than predicted for a smooth flat plate, and the boundary layer may be fully turbulent (or nearly so) by the trailing edge of the fin.

---

## 10-72

**Solution** We are to assess whether the boundary layer on the surface of a sign is laminar or turbulent or transitional.

***Assumptions*** **1** The flow is steady and incompressible. **2** The sign surface is smooth.

***Properties*** The density and viscosity of air at $T$ = 25°C are 1.184 kg/m$^3$ and 1.849 × 10$^{-5}$ kg/m·s respectively. The kinematic viscosity is thus $\nu$ = 1.562 × 10$^{-5}$ m$^2$/s.

***Analysis*** Since the air flow is parallel to the sign, this flow is that of a flat plate boundary layer. We calculate the Reynolds number at the downstream edge of the sign, using $W$ as the streamwise distance along the flat plate,

$$\mathrm{Re}_x = \frac{VW}{\nu} = \frac{(5.0 \text{ m/s})(0.45 \text{ m})}{1.562\times10^{-5} \text{ m}^2/\text{s}} = 1.44\times10^5 \qquad (1)$$

The critical Reynolds number for transition to turbulence is 1 × 10$^5$ for the case of a smooth flat plate with very clean, low-noise freestream conditions. Our Reynolds number is higher than this, but just barely so. The engineering value of critical Reynolds number for transition to a turbulent boundary layer in real engineering flows is $\mathrm{Re}_{x,cr}$ = 5 × 10$^5$; our value of $\mathrm{Re}_x$ is less than $\mathrm{Re}_{x,cr}$. Since $\mathrm{Re}_x$ is a bit greater than $\mathrm{Re}_{x,critical}$, but less than $\mathrm{Re}_{x,cr}$ (5 × 10$^5$), and much less than $\mathrm{Re}_{x,transition}$ (30 × 10$^5$), **the boundary layer is laminar for a while, and then becomes transitional by the trailing edge of the fin**.

***Discussion*** The flow over the sign is not very "clean" – there are eddies from the passing vehicles, and other atmospheric disturbances. In addition, the sign surface is not perfectly smooth, and most signs tend to oscillate somewhat in the wind. Thus, transition and turbulence are likely to occur much earlier than predicted for a smooth flat plate. The boundary layer on this sign is definitely transitional, but probably not turbulent, by the downstream edge of the sign.

---

**10-73E**

**Solution** We are to assess whether the boundary layer on the wall of a wind tunnel is laminar or turbulent or transitional.

***Assumptions*** **1** The flow is steady and incompressible. **2** The surface of the wind tunnel is smooth. **3** There are minimal disturbances in the freestream flow.

***Properties*** The density and viscosity of air at $T = 80^{o}F$ are 0.07350 lbm/ft$^3$ and $1.247 \times 10^{-5}$ lbm/ft·s respectively. The kinematic viscosity is thus $\nu = 1.697 \times 10^{-4}$ ft$^2$/s.

***Analysis*** We calculate the Reynolds number at the downstream end of the wall, using $L = 1.5$ ft as the streamwise distance along the flat plate,

$$\mathrm{Re}_x = \frac{VL}{\nu} = \frac{(7.5\ \text{ft/s})(1.5\ \text{ft})}{1.697\times10^{-4}\ \text{ft}^2/\text{s}} = 6.63\times10^4 \qquad (1)$$

The engineering value of critical Reynolds number for transition to a turbulent boundary layer in real engineering flows is $\mathrm{Re}_{x,cr} = 5 \times 10^5$; our value of $\mathrm{Re}_x$ is much less than $\mathrm{Re}_{x,cr}$. In fact, our Reynolds number is even lower than the critical Reynolds number for transition to turbulence ($1 \times 10^5$) for the case of a smooth flat plate with very clean, low-noise freestream conditions. Since the flow is clean and $\mathrm{Re}_x$ is less than $\mathrm{Re}_{x,critical}$, **the boundary layer is definitely laminar**.

***Discussion*** There is typically a contraction just upstream of the test section of a wind tunnel. Upstream of that are typically some screens and/or honeycombs to make the flow clean and uniform. Thus, the disturbances are likely to be quite small, and the boundary layer is most likely laminar.

---

**10-74**

**Solution** We are to generate an expression for the outer flow velocity at point 2 in the boundary layer.

***Assumptions*** **1** The flow is steady, incompressible, and laminar. **2** The boundary layer approximation is appropriate.

***Analysis*** The boundary layer approximation tells us that $P$ is constant *normal* to the boundary layer, but not necessarily *along* the boundary layer. Therefore, at any streamwise location along the boundary layer, the pressure in the outer flow region just above the boundary layer is the same as that at the wall. In the outer flow region, the Bernoulli equation reduces to

*Outer flow*:
$$U\frac{dU}{dx} = -\frac{1}{\rho}\frac{dP}{dx} \quad \rightarrow \quad \frac{dU}{dx} = -\frac{1}{\rho U}\frac{dP}{dx} \qquad (1)$$

For small values of $\Delta x$, we can approximate $U_2 \approx U_1 + (dU/dx)\Delta x$, and $P_2 \approx P_1 + (dP/dx)\Delta x$. Substitution of these approximations into Eq. 1 yields

$$U_2 \approx U_1 - \frac{1}{\rho U_1}\frac{dP}{dx}\Delta x = U_1 - \frac{1}{\rho U_1}\frac{P_2 - P_1}{\Delta x}\Delta x \quad \rightarrow \quad \boxed{U_2 \approx U_1 - \frac{P_2 - P_1}{\rho U_1}} \qquad (2)$$

***Discussion*** It turns out that $U_2$ does not depend on $\Delta x$ or $\mu$, but only on $P_1$, $P_2$, $U_1$, and $\rho$.

---

**10-75**

**Solution** We are to estimate $U_2$, and explain whether it is less than, equal to, or greater than $U_1$.

***Assumptions*** **1** The flow is steady, incompressible, and laminar. **2** The boundary layer approximation is appropriate.

***Properties*** The density and viscosity of air at $T$ = 25°C are 1.184 kg/m³ and 1.849 × 10⁻⁵ kg/m·s respectively.

***Analysis*** The Bernoulli equation is valid in the outer flow region. Thus, we know that as $P$ increases, $U$ decreases, and vice-versa. In this case, $P$ increases, and thus **we expect $U_2$ to be less than $U_1$**. From the results of Problem 10-74,

$$U_2 \approx U_1 - \frac{P_2 - P_1}{\rho U_1} = 10.3\frac{\text{m}}{\text{s}} - \frac{2.44\ \text{N/m}^2}{\left(1.186\ \text{kg/m}^3\right)\left(10.3\ \text{m/s}\right)}\left(\frac{\text{kg m}}{\text{N s}^2}\right) = \mathbf{10.1\ m/s}$$

Thus, the outer flow velocity indeed decreases by a small amount.

***Discussion*** The approximation is first order, and thus appropriate only if the distance between $x_1$ and $x_2$ is small.

---

**10-76**

**Solution** We are to list the five steps of the boundary layer procedure.

***Analysis*** **We list the five steps below**, with a description of each:

**Step 1** Solve for the outer flow, ignoring the boundary layer (assuming that the region of flow outside the boundary layer is approximately inviscid and/or irrotational). Transform coordinates as necessary to obtain $U(x)$.

**Step 2** Assume a thin boundary layer – so thin in fact that it does not affect the outer flow solution of Step 1.

**Step 3** Solve the boundary layer equations. For this step we use the no-slip boundary condition at the wall, $u = v = 0$ at $y = 0$, the known outer flow condition at the edge of the boundary layer, $u \to U(x)$ as $y \to \infty$, and some known starting profile, $u = u_{\text{starting}}(y)$ at $x = x_{\text{starting}}$.

**Step 4** Calculate quantities of interest in the flow field. For example, once the boundary layer equations have been solved (Step 3), we can calculate $\delta(x)$, shear stress along the wall, total skin friction drag, etc.

**Step 5** Verify that the boundary layer approximations are appropriate. In other words, verify that the boundary layer is indeed *thin* – otherwise the approximation is not justified.

***Discussion*** Students' discussions should be in their own words.

---

**10-77**

**Solution** We are to list at least three "red flags" to look for when performing boundary layer calculations.

***Analysis*** **We list four below**. (Students are asked to list at least three.)

- The boundary layer approximation breaks down if Reynolds number is not large enough. For example, $\delta/L \sim 0.01$ (1%) for $\text{Re}_L = 10{,}000$.
- The assumption of zero pressure gradient in the $y$ direction breaks down if wall curvature is of similar magnitude as $\delta$. In such cases, centripetal acceleration effects due to streamline curvature cannot be ignored. Physically, the boundary layer is not "thin" enough for the approximation to be appropriate when $\delta$ is not $<< R$.
- When Reynolds number is too *high*, the boundary layer does not remain laminar. The boundary layer approximation itself may still be appropriate, but the laminar boundary layer equations are *not* valid if the flow is transitional or fully turbulent. The laminar boundary layer on a smooth flat plate under clean flow conditions begins to transition towards turbulence at $\text{Re}_x \approx 1 \times 10^5$. In practical engineering applications, walls may not be smooth and there may be vibrations, noise, and fluctuations in the freestream flow above the wall, all of which contribute to an even earlier start of the transition process.
- If flow separation occurs, the boundary layer approximation is no longer appropriate in the separated flow region. The main reason for this is that a separated flow region contains *reverse flow*, and the parabolic nature of the boundary layer equations is lost.

***Discussion*** Students' discussions should be in their own words.

---

## 10-78

**Solution** We are to prove that $\tau_w = 0.332\frac{\rho U^2}{\sqrt{\text{Re}_x}}$ for a flat plate boundary layer.

***Assumptions*** **1** The flow is steady and incompressible. **2** The Reynolds number is in the range where the Blasius solution is appropriate.

***Analysis*** Equation 4 of Example 10-10 gives the definition of similarity variable $\eta$, which we re-write in terms of $y$ as a function of $\eta$,

*y as a function of* $\eta$:
$$\eta = y\sqrt{\frac{U}{\nu x}} \quad \rightarrow \quad y = \eta\sqrt{\frac{\nu x}{U}} \tag{1}$$

From the chain rule and Eq. 1, we obtain an expression for $d/d\eta$,

*Derivative with respect to similarity variable* $\eta$:
$$\frac{d}{d\eta} = \frac{d}{dy}\frac{dy}{d\eta} = \frac{d}{dy}\sqrt{\frac{\nu x}{U}} \tag{2}$$

We apply Eq. 2 above to Eq. 8 of Example 10-10,

$$\left.\frac{d(u/U)}{d\eta}\right)_{\eta=0} = \left.\frac{du}{dy}\right)_{y=0}\sqrt{\frac{\nu x}{U^3}} = 0.332 \tag{3}$$

But by definition, $\tau_w = \mu\left.\frac{\partial u}{\partial y}\right)_{y=0}$, and Eq. 3 yields

*Shear stress at the wall*:
$$\tau_w = \mu\left.\frac{du}{dy}\right)_{y=0}\sqrt{\frac{\nu x}{U^3}} = 0.332\rho\nu\sqrt{\frac{U^3}{\nu x}} = 0.332\rho U^2\sqrt{\frac{\nu}{Ux}} = 0.332\frac{\rho U^2}{\sqrt{\text{Re}_x}} \tag{4}$$

**which is the desired expression for the shear stress at the wall in physical variables**.

***Discussion*** The chain rule algebra is valid here since *U* and *x* are functions of *x* only – they are not functions of *y*.

## 10-79E

**Solution** We are to calculate $\delta$, $\delta^*$, and $\theta$ at the end of the wind tunnel test section.

***Assumptions*** **1** The flow is steady and incompressible. **2** The surface of the wind tunnel is smooth. **3** The boundary layer remains laminar all the way to the end of the test section.

***Properties*** The density and viscosity of air at $T = 80^{\circ}$F are 0.07350 lbm/ft$^3$ and $1.247 \times 10^{-5}$ lbm/ft·s respectively.

***Analysis*** In Problem 10-73E, we calculated the Reynolds number at the downstream end of the wall, $\mathrm{Re}_x = 6.63 \times 10^4$ (keeping an extra digit for the calculations). All of the desired quantities are functions of $\mathrm{Re}_x$:

*Boundary layer thickness*:

$$\delta = \frac{4.91x}{\sqrt{\mathrm{Re}_x}} = \frac{4.91(1.5\text{ ft})}{\sqrt{6.63\times10^4}} = 0.0286\text{ ft} \approx 0.34\text{ in} \quad (1)$$

and

*Displacement thickness*:

$$\delta^* = \frac{1.72x}{\sqrt{\mathrm{Re}_x}} = \frac{1.72(1.5\text{ ft})}{\sqrt{6.63\times10^4}} = 0.0100\text{ ft} \approx 0.12\text{ in} \quad (2)$$

and

*Momentum thickness*:

$$\theta = \frac{0.664x}{\sqrt{\mathrm{Re}_x}} = \frac{0.664(1.5\text{ ft})}{\sqrt{6.63\times10^4}} = 0.00387\text{ ft} \approx 0.046\text{ in} \quad (3)$$

Thus, $\boldsymbol{\delta}$ **= 0.34 inches**, $\boldsymbol{\delta^*}$ **= 0.12 inches**, and $\boldsymbol{\theta}$ **= 0.046 inches** at the end of the wind tunnel test section. As expected, $\delta > \delta^* > \theta$.

***Discussion*** All answers are given to two significant digits.

---

## 10-80

**Solution** We are to determine which orientation of a rectangular flat plate produces the higher drag.

***Assumptions*** **1** The flow is steady and incompressible. **2** The Reynolds number is high enough for a laminar boundary layer to form on the plate, but not high enough for the boundary layer to become turbulent.

***Analysis*** Reynolds number appears in the denominator of the equation for shear stress along the wall of a laminar boundary layer. Thus, wall shear stress decreases with increasing $x$, the distance down the plate. Hence, the average wall shear stress is higher for the case with the plate oriented with its short dimension aligned with the wind (case (b) of Fig. P10-80). Since the surface area of the plate is the same regardless of orientation, the plate with the higher average value of $\tau_w$ has the higher overall drag. **Case (b) has the higher drag**.

***Discussion*** Another way to think about this problem is that since the boundary layer is thinner near the leading edge, the shear stress is higher there, and the front portion of the plate contributes to more of the total drag than does the rear portion of the plate.

---

**10-81**

**Solution** We are to define displacement thickness and discuss whether it is larger or smaller than boundary layer thickness.

***Assumptions*** **1** The flow is steady and incompressible. **2** The boundary layer growing on the flat plate is laminar.

***Analysis*** The two definitions of displacement thickness are:

- Displacement thickness is the distance that a streamline just outside of the boundary layer is deflected away from the wall due to the effect of the boundary layer.
- Displacement thickness is the imaginary increase in thickness of the wall, as seen by the outer flow, due to the effect of the growing boundary layer.

For a laminar boundary layer, **$\delta$ is larger than $\delta^*$**. $\delta$ is defined by the overall thickness of the boundary layer, whereas $\delta^*$ is an integrated thickness across the boundary layer that averages the mass deficit across the boundary layer. Therefore, it is not surprising that $\delta^*$ is less than $\delta$.

***Discussion*** The definitions given by students should be in their own words.

**10-82**

**Solution** The acceleration of air through the round test section of a wind tunnel is to be calculated.

***Assumptions*** **1** The flow is steady and incompressible. **2** The walls are smooth, and disturbances and vibrations are kept to a minimum. **3** The boundary layers is laminar.

***Properties*** The kinematic viscosity of air at 20°C is $\nu = 1.516 \times 10^{-5}$ m$^2$/s.

***Analysis*** (*a*) As the boundary layer grows along the wall of the wind tunnel test section, air in the region of irrotational flow in the central portion of the test section accelerates in order to satisfy conservation of mass. The Reynolds number at the end of the test section is

$$\text{Re}_x = \frac{Vx}{\nu} = \frac{(2.0 \text{ m/s})(0.60 \text{ m})}{1.516\times10^{-5} \text{ m}^2/\text{s}} = 7.92\times10^4$$

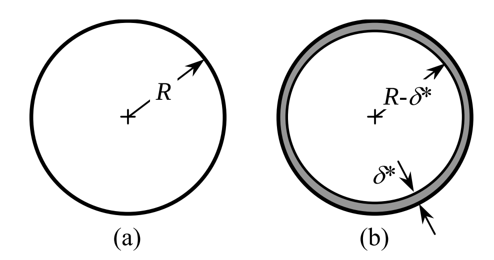



**FIGURE 1**
Cross-sectional views of the test section of the wind tunnel: (a) beginning of test section, and (b) end of test section.

The engineering value of critical Reynolds number for transition to a turbulent boundary layer in real engineering flows is $\text{Re}_{x,\text{cr}} = 5 \times 10^5$; our value of $\text{Re}_x$ is less than $\text{Re}_{x,\text{cr}}$. In fact, $\text{Re}_x$ is lower than the critical Reynolds number, $\text{Re}_{x,\text{critical}} \approx 1 \times 10^5$, for a smooth flat plate with a clean free stream. Since the walls are smooth and the flow is clean, we may assume that the boundary layer on the wall remains laminar throughout the length of the test section. We estimate the displacement thickness at the end of the test section,

$$\delta^* \approx \frac{1.72x}{\sqrt{\text{Re}_x}} = \frac{1.72(0.60 \text{ m})}{\sqrt{7.92\times10^4}} = 3.67\times10^{-3} \text{ m} = 3.67 \text{ mm} \qquad (1)$$

Two cross-sectional views of the test section are sketched in Fig. 1, one at the beginning and one at the end of the test section. The effective radius at the end of the test section is reduced by $\delta^*$ as calculated by Eq. 1. We apply conservation of mass to calculate the air speed at the end of the test section,

$$V_{\text{end}}A_{\text{end}} = V_{\text{beginning}}A_{\text{beginning}} \quad \rightarrow \quad V_{\text{end}} = V_{\text{beginning}}\frac{\pi R^2}{\pi(R-\delta^*)^2} \qquad (2)$$

We plug in the numerical values to obtain

*Result*:
$$V_{\text{end}} = (2.0 \text{ m/s})\frac{(0.20 \text{ m})^2}{(0.20 \text{ m} - 3.67\times10^{-3} \text{ m})^2} = \mathbf{2.08 \text{ m/s}} \qquad (3)$$

Thus the air speed increases by approximately 4% through the test section, due to the effect of displacement thickness.

***Discussion*** The same displacement thickness technique may be applied to turbulent boundary layers; however, a different equation for $\delta^*(x)$ is required.

**10-83**

**Solution** The acceleration of air through the square test section of a wind tunnel is to be calculated and compared to that through a round wind tunnel test section.

***Assumptions*** **1** The flow is steady and incompressible. **2** The walls are smooth, and disturbances and vibrations are kept to a minimum. **3** The boundary layers is laminar.

***Properties*** The kinematic viscosity of air at 20°C is $\nu = 1.516 \times 10^{-5}$ m²/s.

***Analysis*** (*a*) As the boundary layer grows along the walls of the wind tunnel test section, air in the region of irrotational flow in the central portion of the test section accelerates in order to satisfy conservation of mass. The Reynolds number at the end of the test section is

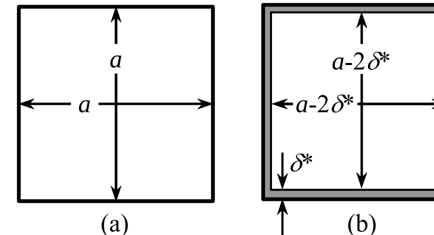



**FIGURE 1**
Cross-sectional views of the test section of the wind tunnel: (a) beginning of test section, and (b) end of test section.

$$\text{Re}_x = \frac{Vx}{\nu} = \frac{(2.0 \text{ m/s})(0.60 \text{ m})}{1.516\times10^{-5}\text{m}^2/\text{s}} = 7.92\times10^4$$

The engineering value of critical Reynolds number for transition to a turbulent boundary layer in real engineering flows is $\text{Re}_{x,cr} = 5 \times 10^5$; our value of $\text{Re}_x$ is less than $\text{Re}_{x,cr}$. In fact, $\text{Re}_x$ is lower than the critical Reynolds number, $\text{Re}_{x,\text{critical}} \approx 1 \times 10^5$, for a smooth flat plate with a clean free stream. Since the walls are smooth and the flow is clean, we may assume that the boundary layer on the wall remains laminar throughout the length of the test section. We estimate the displacement thickness at the end of the test section,

$$\delta^* \approx \frac{1.72x}{\sqrt{\text{Re}_x}} = \frac{1.72(0.60 \text{ m})}{\sqrt{7.92\times10^4}} = 3.67\times10^{-3} \text{ m} = 3.67 \text{ mm} \qquad (1)$$

Two cross-sectional views of the test section are sketched in Fig. 1, one at the beginning and one at the end of the test section. The effective dimensions at the end of the test section are reduced by $2\delta^*$. We apply conservation of mass to calculate the air speed at the end of the test section,

$$V_{\text{end}}A_{\text{end}} = V_{\text{beginning}}A_{\text{beginning}} \quad \rightarrow \quad V_{\text{end}} = V_{\text{beginning}}\frac{a^2}{(a-2\delta^*)^2} \qquad (2)$$

We plug in the numerical values to obtain

*Result*:
$$V_{\text{end}} = (2.0 \text{ m/s})\frac{(0.40 \text{ m})^2}{\left[0.40 \text{ m} - 2(3.67\times10^{-3} \text{ m})\right]^2} = \mathbf{2.08\,m/s} \qquad (3)$$

Thus the air speed increases by approximately 4% through the test section, due to the effect of displacement thickness.

The result for the square test section is identical to that of the round test section. We might have expected the square test section to do better since its cross-sectional area is larger than that of the round test section. However, the square test section also has more wall surface area than does the round test section, and thus, the acceleration due to displacement thickness on the walls is the same in both cases.

***Discussion*** The same displacement thickness technique may be applied to turbulent boundary layers; however, a different equation for $\delta^*(x)$ is required.

**10-84**

**Solution** The height of a boundary layer scoop in a wind tunnel test section is to be calculated.

***Assumptions*** **1** The flow is steady and incompressible. **2** The walls are smooth. **3** The boundary layers starts growing at $x = 0$.

***Properties*** The kinematic viscosity of air at 20°C is $\nu = 1.516 \times 10^{-5}$ m²/s.

***Analysis*** (*a*) As the boundary layer grows along the wall of the wind tunnel test section, the Reynolds number increases. The Reynolds number at location *x* is

*Re$_x$*:
$$\text{Re}_x = \frac{Vx}{\nu} = \frac{(65.0 \text{ m/s})(1.45 \text{ m})}{1.516\times10^{-5}\text{ m}^2/\text{s}} = 6.22\times10^6$$

Since $\text{Re}_x$ is greater than the transition Reynolds number, $\text{Re}_{x,\text{transition}} \approx 3 \times 10^6$, we assume that the boundary layer is turbulent throughout the length of the test section. We estimate the boundary layer thickness at the location of the scoop,

*Table 10-4a*:
$$\delta \approx \frac{0.16x}{(\text{Re}_x)^{1/7}} = \frac{0.16(1.45 \text{ m})}{(6.22\times10^6)^{1/7}} = 2.48\times10^{-2} \text{ m} = 24.8 \text{ mm} \quad (1)$$

or,

*Table 10-4b*:
$$\delta \approx \frac{0.38x}{(\text{Re}_x)^{1/5}} = \frac{0.38(1.45 \text{ m})}{(6.22\times10^6)^{1/5}} = 2.41\times10^{-2} \text{ m} = 24.1 \text{ mm} \quad (1)$$

We design the scoop height to be greater than or equal to the boundary layer thickness at the location of the scoop. Thus, we set **$h \approx \delta \approx$ 25. mm, or about an inch**.

***Discussion*** The suction pressure of the scoop must be adjusted carefully so as not to suck too much or too little – otherwise it would disturb the flow. Since the early portion of the boundary layer is laminar, the actual boundary layer thickness will be somewhat lower than that calculated here. Thus, our calculation represents an upper limit. However, some of the large turbulent eddies in the boundary layer may actually exceed height $\delta$, so our calculated *h* may actually not be sufficient to remove the complete boundary layer.

**10-85E**

**Solution** The acceleration of air through the round test section of a wind tunnel is to be calculated.

***Assumptions*** **1** The flow is steady and incompressible. **2** The walls are smooth, and disturbances and vibrations are kept to a minimum. **3** The boundary layers is laminar.

***Properties*** The kinematic viscosity of air at 70°F is $\nu = 1.643 \times 10^{-4}$ ft²/s.

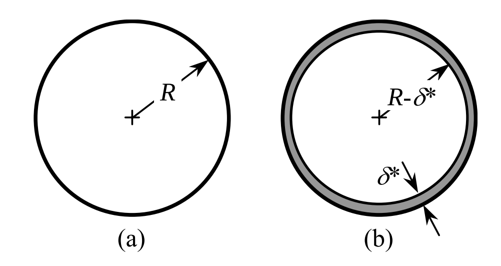



**FIGURE 1**
Cross-sectional views of the test section of the wind tunnel: (a) beginning of test section, and (b) end of test section.

***Analysis*** (*a*) As the boundary layer grows along the wall of the wind tunnel test section, air in the region of irrotational flow in the central portion of the test section accelerates in order to satisfy conservation of mass. The Reynolds number at the end of the test section is

$$\text{Re}_x = \frac{Vx}{\nu} = \frac{(5.0\text{ ft/s})(10.0\text{ in})}{1.643\times10^{-4}\text{ ft}^2\text{/s}}\left(\frac{1\text{ft}}{12\text{ in}}\right) = 2.54\times10^4$$

The engineering value of critical Reynolds number for transition to a turbulent boundary layer in real engineering flows is $\text{Re}_{x,cr} = 5 \times 10^5$; our value of $\text{Re}_x$ is less than $\text{Re}_{x,cr}$. In fact, $\text{Re}_x$ is lower than the critical Reynolds number, $\text{Re}_{x,\text{critical}} \approx 1 \times 10^5$, for a smooth flat plate with a clean free stream. Since the walls are smooth and the flow is clean, we may assume that the boundary layer on the wall remains laminar throughout the length of the test section. We estimate the displacement thickness at the end of the test section,

$$\delta^* \approx \frac{1.72x}{\sqrt{\text{Re}_x}} = \frac{1.72(10.0\text{ in})}{\sqrt{2.54\times10^4}} = 0.108\text{ in} \qquad (1)$$

Two cross-sectional views of the test section are sketched in Fig. 1, one at the beginning and one at the end of the test section. The effective radius at the end of the test section is reduced by $\delta^*$ as calculated by Eq. 1. We apply conservation of mass to calculate the air speed at the end of the test section,

$$V_{\text{end}}A_{\text{end}} = V_{\text{beginning}}A_{\text{beginning}} \quad \rightarrow \quad V_{\text{end}} = V_{\text{beginning}}\frac{\pi R^2}{\pi(R-\delta^*)^2} \qquad (2)$$

We plug in the numerical values to obtain

*Result*:
$$V_{\text{end}} = (5.0\text{ ft/s})\frac{(6.0\text{ in})^2}{(6.0\text{ in}-0.108\text{ in})^2} = \mathbf{5.18\ ft/s} \qquad (3)$$

Thus the air speed increases by approximately 4% through the test section, due to the effect of displacement thickness.

**To eliminate this acceleration, the engineers can either diverge the test section walls, or add some suction along the sides to remove some air**.

***Discussion*** The same displacement thickness technique may be applied to turbulent boundary layers; however, a different equation for $\delta^*(x)$ is required.

**10-86E**

**Solution** We are to determine if a boundary layer is laminar, turbulent, or transitional, and then compare the laminar and turbulent boundary layer thicknesses.

***Properties*** The kinematic viscosity of air at 70°F is $\nu = 1.643 \times 10^{-4}$ ft²/s.

***Analysis*** First, we calculate the Reynolds number at the end of the plate,

*Re$_x$, end of plate*:
$$\mathrm{Re}_x = \frac{Vx}{\nu} = \frac{(15.5 \text{ ft/s})(10.6 \text{ ft})}{1.643\times10^{-4}\text{ ft}^2\text{/s}} = 1.00\times10^6$$

The engineering value of critical Reynolds number for transition to a turbulent boundary layer in real engineering flows is $\mathrm{Re}_{x,cr} = 5 \times 10^5$; our value of $\mathrm{Re}_x$ is greater than $\mathrm{Re}_{x,cr}$, leading us to suspect that the boundary layer is turbulent. However, $\mathrm{Re}_x$ is lower than the transition Reynolds number, $\mathrm{Re}_{x,\text{transition}} \approx 3 \times 10^6$, for a smooth flat plate with a clean free stream. Thus, we suspect that **this boundary layer is laminar at the front of the plate, and then transitional farther downstream**. If the plate is vibrating and/or the freestream is noisy, the boundary layer may possibly be fully turbulent by the end of the plate.

If the boundary layer were to remain laminar to the end of the plate, its thickness would be

*Laminar*:
$$\delta = \frac{4.91x}{\sqrt{\mathrm{Re}_x}} = \frac{4.91(10.6 \text{ ft})}{\sqrt{1.00\times10^6}} = 0.0520 \text{ ft} = \mathbf{0.625\ in} \qquad (1)$$

If the boundary layer at the end of the plate were fully turbulent (and turbulent from the beginning of the plate), its thickness would be

*Turbulent, Table 10-4a*:
$$\delta \approx \frac{0.16x}{(\mathrm{Re}_x)^{1/7}} = \frac{0.16(10.6 \text{ ft})}{(1.00\times10^6)^{1/7}} = 0.236 \text{ ft} = \mathbf{2.83\ in} \qquad (1)$$

or,

*Turbulent, Table 10-4b*:
$$\delta \approx \frac{0.38x}{(\mathrm{Re}_x)^{1/5}} = \frac{0.38(10.6 \text{ ft})}{(1.00\times10^6)^{1/5}} = 0.254 \text{ ft} = \mathbf{3.05\ in} \qquad (1)$$

Thus, the turbulent boundary layer thickness is about 4.5 to 4.9 times thicker than the corresponding laminar boundary layer thickness at the same Reynolds number. We expect the actual boundary layer thickness to lie somewhere between these two extremes.

***Discussion*** The difference between the two turbulent boundary layer equations for $\delta$ is about 7 or 8 percent. This is larger than we might hope, but keep in mind that at $\mathrm{Re}_x = 1.00 \times 10^6$, the boundary layer is not yet fully turbulent, and the equations for $\delta$ are not accurate at such low Reynolds numbers.

**10-87**

**Solution** The apparent thickness of a flat plate is to be calculated.

***Assumptions*** **1** The flow is steady and incompressible. **2** The walls are smooth. **3** The boundary layers starts growing at $x = 0$.

***Properties*** The kinematic viscosity of air at 20°C is $\nu = 1.516 \times 10^{-5}$ m²/s.

***Analysis*** (*a*) As the boundary layer grows along the plate, the Reynolds number increases. The Reynolds number at location $x$ is

$Re_x$:
$$\text{Re}_x = \frac{Vx}{\nu} = \frac{(5.0 \text{ m/s})(0.25 \text{ m})}{1.516\times10^{-5}\text{m}^2/\text{s}} = 8.2454\times10^4$$

The engineering value of critical Reynolds number for transition to a turbulent boundary layer in real engineering flows is $\text{Re}_{x,cr} = 5 \times 10^5$; our value of $\text{Re}_x$ is less than $\text{Re}_{x,cr}$. In fact, $\text{Re}_x$ is lower than the critical Reynolds number, $\text{Re}_{x,\text{critical}} \approx 1 \times 10^5$, for a smooth flat plate with a clean free stream. Since $\text{Re}_x$ is lower than the critical Reynolds number, and since the walls are smooth and the flow is clean, we may assume that the boundary layer on the wall remains laminar, at least to location $x$. We estimate the displacement thickness at $x = 25$ cm,

$$\delta^* \approx \frac{1.72x}{\sqrt{\text{Re}_x}} = \frac{1.72(0.25 \text{ m})}{\sqrt{8.2454\times10^4}} = 1.4975\times10^{-3} \text{ m} = 0.14975 \text{ cm} \qquad (1)$$

This "extra" thickness is seen by the outer flow. Since the plate is 0.75 cm thick, and since a similar boundary layer forms on the bottom as on the top, the total apparent thickness of the plate is

*Apparent thickness*: $h_{\text{apparent}} = h + 2\delta^* = 0.75 \text{ cm} + 2(0.14975 \text{ cm}) = \mathbf{1.05\ cm}$

***Discussion*** We have kept 5 digits of precision in intermediate steps, but report our final answer to three significant digits. The Reynolds number is pretty close to critical. If the freestream air flow were noisy and/or the plate were rough or vibrating, we might expect the boundary layer to be transitional, and then the apparent thickness would be greater.

**10-88** [*Also solved using EES on enclosed DVD*]

**Solution** We are to plot the mean boundary layer profile $u(y)$ at the end of a flat plate using three different approximations.

***Assumptions*** **1** The plate is smooth. **2** The boundary layer is turbulent from the beginning of the plate. **3** The flow is steady in the mean. **4** The plate is infinitesimally thin and is aligned parallel to the freestream.

***Properties*** The kinematic viscosity of air at 20°C is $\nu = 1.516 \times 10^{-5}$ m²/s.

***Analysis*** First we calculate the Reynolds number at $x = L$,

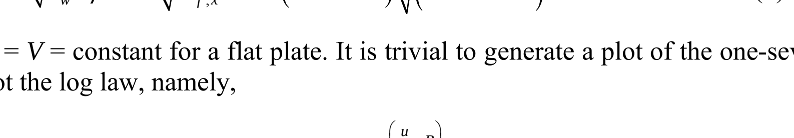
$$\text{Re}_x = \frac{Vx}{\nu} = \frac{(80.0\text{ m/s})(17.5\text{ m})}{1.516\times10^{-5}\text{ m}^2/\text{s}} = 9.23\times10^7$$

This value of $\text{Re}_x$ is well above the transitional Reynolds number for a flat plate boundary layer, so the assumption of turbulent flow from the beginning of the plate is reasonable.

Using the column (a) values of Table 10-4, we estimate the boundary layer thickness and the local skin friction coefficient at the end of the plate,

$$\delta \approx \frac{0.16x}{(\text{Re}_x)^{1/7}} = 0.204\text{ m} \qquad C_{f,x} \approx \frac{0.027}{(\text{Re}_x)^{1/7}} = 1.97\times10^{-3} \qquad (1)$$

We calculate the friction velocity by using the definition of $C_{f,x}$,



$$u_* = \sqrt{\tau_w/\rho} = U\sqrt{C_{f,x}/2} = (80.0\text{ m/s})\sqrt{(1.97\times10^{-3})/2} = 2.51\text{ m/s} \qquad (2)$$



where $U(x) = V$ = constant for a flat plate. It is trivial to generate a plot of the one-seventh-power law. We follow Example 10-13 to plot the log law, namely,

$$y = \frac{\nu}{u_*}e^{\kappa\left(\frac{u}{u_*}-B\right)} \qquad (3)$$

Since we know that $u$ varies from 0 at the wall to $U$ at the boundary layer edge, we are also able to plot the log law velocity profile in physical variables. Finally, Spalding's law of the wall is also written in terms of $y$ as a function of $u$. **We plot all three profiles on the same plot for comparison (Fig. 1)**. All three are close, and we cannot distinguish the log law from Spalding's law on this scale.

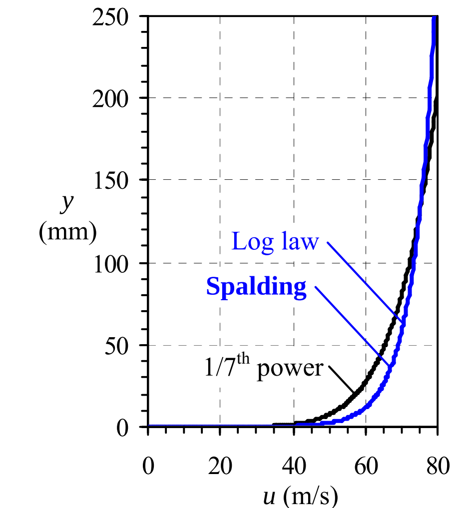



**FIGURE 1**
Comparison of turbulent flat plate boundary layer profile expressions in physical variables at $\text{Re}_x = 9.23 \times 10^7$: one-seventh-power approximation, log law, and Spalding's law of the wall.

***Discussion*** Neither the one-seventh-power law nor the log law are valid real close to the wall, but Spalding's law is valid all the way to the wall. However, on the scale shown in Fig. 1, we cannot see the differences between the log law and the Spalding law very close to the wall.

**10-89**

**Solution** We are to discuss the difference between a favorable and an adverse pressure gradient.

***Analysis*** **When the pressure decreases downstream, the boundary layer is said to experience to a favorable pressure gradient. When the pressure increases downstream, the boundary layer is subjected to an adverse pressure gradient.** The term "favorable" is used because the boundary layer is unlikely to separate off the wall. On the other hand, "adverse" or "unfavorable" indicates that the boundary layer is more likely to separate off the wall.

***Discussion*** A favorable pressure gradient occurs typically at the front of a body, whereas an adverse pressure gradient occurs typically at the back portion of a body.

**10-90**

**Solution** We are to discuss the role of an inflection point in a boundary layer profile.

***Analysis*** As sketched in Fig. 10-124, **the existence of an inflection point in the boundary layer profile indicates an adverse or unfavorable pressure gradient**. The reason for this is due to the fact that the second derivative of the velocity profile $u(y)$ at the wall is directly proportional to the pressure gradient (Eq. 10-86). In an adverse pressure gradient field, $dP/dx$ is *positive*, and thus, $\partial^2 u/\partial y^2)_{y=0}$ is also *positive*. However, since $\partial^2 u/\partial y^2$ must be *negative* as $u$ approaches $U(x)$ at the edge of the boundary layer, there has to be an *inflection point* ($\partial^2 u/\partial y^2 = 0$) somewhere in the boundary layer.

***Discussion*** If the adverse pressure gradient is large enough, the boundary layer separates off the wall, leading to reverse flow near the wall.

---

**10-91**

**Solution** We are to compare laminar and turbulent boundary layer separation, and explain why golf balls have dimples.

***Analysis*** **Turbulent boundary layers are more "full" than are laminar boundary layers**. Because of this, **a turbulent boundary layer is much less likely to separate compared to a laminar boundary layer under the same adverse pressure gradient**. A smooth golf ball, for example, would maintain a laminar boundary layer on its surface, and the boundary layer would separate fairly easily, leading to large aerodynamic drag. **Golf balls have dimples (a type of surface roughness) in order to create an early transition to a turbulent boundary layer**. Flow still separates from the golf ball surface, but much farther downstream in the boundary layer, resulting in significantly reduced aerodynamic drag.

***Discussion*** Turbulent boundary layers have more skin friction drag than do laminar boundary layers, but this effect is less significant than the pressure drag caused by flow separation. Thus, a rough golf ball (at appropriate Reynolds numbers) ends up with less overall drag, compared to a smooth golf ball at the same conditions.

---

**10-92**

**Solution** We are to generate expressions for $\delta^*$ and $\theta$, and compare to Blasius.

***Analysis*** First, we set $U(x) = V$ = constant for a flat plate. We integrate using the definition of $\delta^*$,

$$\delta^* = \int_0^\infty \left(1-\frac{u}{U}\right)dy = \int_0^\delta \left[1-\frac{y}{\delta}\right]dy = \left[y-\frac{y^2}{2\delta}\right]_{y=0}^{y=\delta}$$

We integrate only to $y = \delta$, since beyond that, the integrand is identically zero. After substituting the limits of integration, we obtain $\delta^*$ as a function of $\delta$,

$$\delta^* = \left[\delta-\delta/2\right]-\left[0-0\right] = \delta/2 \quad \rightarrow \quad \boldsymbol{\delta^* = \delta/2}$$

Similarly,

$$\theta = \int_0^\infty \frac{u}{U}\left(1-\frac{u}{U}\right)dy = \int_0^\delta \left[\frac{y}{\delta}-\frac{y^2}{\delta^2}\right]dy = \left[\frac{y^2}{2\delta}-\frac{y^3}{3\delta^2}\right]_{y=0}^{y=\delta}$$

After substituting the limits of integration, we obtain $\theta$ as a function of $\delta$,

$$\theta = \left[\delta/2-\delta/3\right]-\left[0-0\right] = \delta/6 \quad \rightarrow \quad \boldsymbol{\theta = \delta/6}$$

The ratios are $\boldsymbol{\delta^*/\delta}$ **= 1/2 = 0.500**, and $\boldsymbol{\theta/\delta}$ **= 1/6 = 0.167**, to three significant digits. We compare these approximate results to those obtained from the Blasius solution, i.e., $\delta^*/\delta = 1.72/4.91 = 0.350$, and $\theta/\delta = 0.664/4.91 = 0.135$. Thus, **our approximate velocity profile yields** $\boldsymbol{\delta^*/\delta}$ **to about 43% error, and** $\boldsymbol{\theta/\delta}$ **to about 23% error**.

***Discussion*** The linear approximation is not very accurate.

---

## 10-93

**Solution** We are to generate an expression for $\delta/x$, and compare to Blasius.

***Analysis*** By definition of local skin friction coefficient $C_{f,x}$,

$$C_{f,x} = \frac{2\tau_w}{\rho U^2} = \frac{2}{\rho U^2}\left(\mu \frac{du}{dy}\right)_{y=0} = \frac{2}{\rho U^2}\left[\mu \frac{U}{\delta}\right]_{y=0} = \frac{2\mu}{\rho U \delta} \qquad (1)$$

For a flat plate, the Kármán integral equation reduces to

$$C_{f,x} = 2\frac{d\theta}{dx} = 2\left(\frac{1}{6}\right)\frac{d\delta}{dx} \qquad (2)$$

where we have used the expression for $\theta$ as a function of $\delta$ from Problem 10-92. Substitution of Eq. 1 into Eq. 2 gives

$$\frac{d\delta}{dx} = 3C_{f,x} = \frac{6\mu}{\rho U \delta}$$

We separate variables and integrate,

$$\delta d\delta = \frac{6\mu}{\rho U}dx \quad \rightarrow \quad \frac{\delta^2}{2} = \frac{6\mu}{\rho U}x \quad \rightarrow \quad \frac{\delta}{x} = \sqrt{12\frac{\mu}{\rho U x}}$$

or, collecting terms, rounding to three digits, and setting $\rho Ux/\mu = \mathrm{Re}_x$,

$$\boxed{\frac{\delta}{x} = \frac{3.46}{\sqrt{\mathrm{Re}_x}}}$$

Compared to the Blasius result, $\delta/x = 4.91/\sqrt{\mathrm{Re}_x}$, our approximation based on the sine function velocity profile yields less than 30% error.

***Discussion*** The Kármán integral equation is useful for obtaining approximate relations, and is "forgiving" because of the integration. Even so, the linear approximation is not very good. Nevertheless, a 30% error is sometimes reasonable for "back of the envelope" calculations. The sine wave approximation does much better, as in the next problem.

---

## 10-94



**Solution** We are to compare the sine wave approximation to the Blasius velocity profile.

***Analysis*** We plot both profiles in Fig. 1. There is not much difference between the two, and thus, **the sine wave profile is a very good approximation of the Blasius profile**.

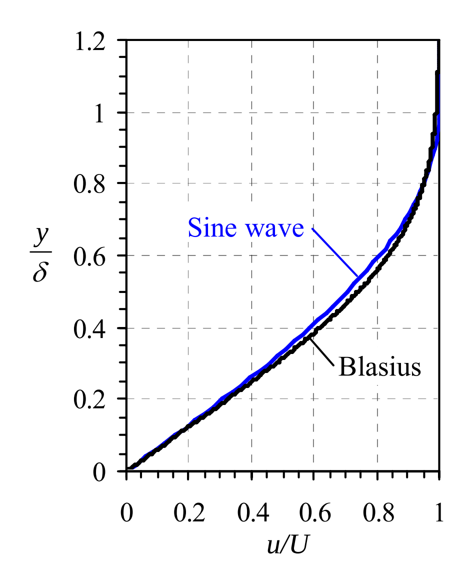



**FIGURE 1**
Comparison of Blasius and sine wave velocity profiles.

***Discussion*** The slope of the two profiles at the wall is indistinguishable on the plot (Fig. 1); thus, the sine wave approximation should yield reasonable results for skin friction (shear stress) along the wall as well.

---

**10-95**

**Solution** We are to generate expressions for $\delta^*$ and $\theta$, and compare to Blasius.

***Analysis*** First, we set $U(x) = V$ = constant for a flat plate. We integrate using the definition of $\delta^*$,

$$\delta^* = \int_0^\infty \left(1-\frac{u}{U}\right)dy = \int_0^\delta \left[1-\sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)\right]dy = \left[y+\cos\left(\frac{\pi}{2}\frac{y}{\delta}\right)\frac{2\delta}{\pi}\right]_{y=0}^{y=\delta}$$

We integrate only to $y = \delta$, since beyond that, the integrand is identically zero. After substituting the limits of integration, we obtain $\delta^*$ as a function of $\delta$,

$$\delta^* = \left[\delta+0\right]-\left[0+\frac{2\delta}{\pi}\right] = \delta - \frac{2}{\pi}\delta \quad \rightarrow \quad \boldsymbol{\delta^* = 0.3634\delta}$$

Similarly,

$$\theta = \int_0^\infty \frac{u}{U}\left(1-\frac{u}{U}\right)dy = \int_0^\delta \left[\sin\left(\frac{\pi}{2}\frac{y}{\delta}\right)-\sin^2\left(\frac{\pi}{2}\frac{y}{\delta}\right)\right]dy$$

$$= \left\{-\cos\left(\frac{\pi}{2}\frac{y}{\delta}\right)\frac{2\delta}{\pi}-\left[\frac{y}{2}-\frac{\delta}{2\pi}\sin\left(\pi\frac{y}{\delta}\right)\right]\right\}_{y=0}^{y=\delta}$$

where we obtained the integral for $\sin^2$ from integration tables. After substituting the limits of integration, we obtain $\theta$ as a function of $\delta$,

$$\theta = \left\{0-\left[\frac{\delta}{2}-0\right]\right\}-\left\{-\frac{2\delta}{\pi}-\left[0-0\right]\right\} = -\frac{\delta}{2}+\frac{2\delta}{\pi} \quad \rightarrow \quad \boldsymbol{\theta = 0.1366\delta}$$

The ratios are $\boldsymbol{\delta^*/\delta = 0.363}$, and $\boldsymbol{\theta/\delta = 0.137}$, to three significant digits. We compare these approximate results to those obtained from the Blasius solution, i.e., $\delta^*/\delta$ = 1.72/4.91 = 0.350, and $\theta/\delta$ = 0.664/4.91 = 0.135. Thus, **our approximate velocity profile yields $\boldsymbol{\delta^*/\delta}$ to less than 4% error, and $\boldsymbol{\theta/\delta}$ to about 1% error**.

***Discussion*** Integration is "forgiving", and reasonable results can be obtained from integration, even when the velocity profile shape is not exact.

**10-96**

**Solution** We are to generate an expression for $\delta/x$, and compare to Blasius.

***Analysis*** By definition of local skin friction coefficient $C_{f,x}$,

$$C_{f,x} = \frac{2\tau_w}{\rho U^2} = \frac{2}{\rho U^2}\left(\mu \frac{du}{dy}\right)_{y=0} = \frac{2}{\rho U^2}\left[\mu U \cos\left(\frac{\pi}{2}\frac{y}{\delta}\right)\frac{\pi}{2\delta}\right]_{y=0} = \frac{2\mu}{\rho U}\left[\frac{\pi}{2\delta}\right] = \frac{\mu\pi}{\rho U\delta} \qquad (1)$$

For a flat plate, the Kármán integral equation reduces to

$$C_{f,x} = 2\frac{d\theta}{dx} = 2(0.1366)\frac{d\delta}{dx} \qquad (2)$$

where we have used the expression for $\theta$ as a function of $\delta$ from Problem 10-95. Substitution of Eq. 1 into Eq. 2 gives

$$\frac{d\delta}{dx} = \frac{C_{f,x}}{2(0.1366)} = \frac{\mu\pi}{0.2732\rho U\delta}$$

We separate variables and integrate,

$$\delta d\delta = \frac{\mu\pi}{0.2732\rho U}dx \quad\rightarrow\quad \frac{\delta^2}{2} = \frac{\mu\pi}{0.2732\rho U}x \quad\rightarrow\quad \frac{\delta}{x} = \sqrt{\frac{2\pi}{0.2732}\frac{\mu}{\rho Ux}}$$

Collecting terms, rounding to three digits, and setting $\rho Ux/\mu = \text{Re}_x$,

$$\boxed{\frac{\delta}{x} \approx \frac{4.80}{\sqrt{\text{Re}_x}}}$$

Compared to the Blasius result, $\delta/x = 4.91/\sqrt{\text{Re}_x}$ , **the approximation yields less than 3% error**.

***Discussion*** The Kármán integral equation is useful for approximations, and is “forgiving” because of the integration.

---

**10-97**

**Solution** We are to compare $H$ for laminar vs. turbulent boundary layers, and discuss its significance.

***Analysis*** Shape factor $H$ is defined as the ratio of displacement thickness to momentum thickness. Thus,

*Shape factor*:
$$H = \frac{\delta^*}{\theta} = \frac{\delta^*/x}{\theta/x} \qquad (1)$$

For the laminar boundary layer on a flat plate, Eq. 1 becomes

*Laminar*:
$$H = \frac{\delta^*/x}{\theta/x} = \frac{1.72/\sqrt{\text{Re}_x}}{0.664/\sqrt{\text{Re}_x}} = \mathbf{2.59}$$

For the turbulent boundary layer, using both columns for comparison, Eq. 1 yields

*Table 10-4a*:
$$H = \frac{\delta^*/x}{\theta/x} = \frac{0.020/(\text{Re}_x)^{1/7}}{0.016/(\text{Re}_x)^{1/7}} = \mathbf{1.25}$$

*Table 10-4b*:
$$H = \frac{\delta^*/x}{\theta/x} = \frac{0.048/(\text{Re}_x)^{1/5}}{0.037/(\text{Re}_x)^{1/5}} = \mathbf{1.30}$$

Thus, **the shape factor for a laminar boundary layer is about twice that of turbulent boundary layer**. This implies that the smaller the value of $H$, the more full is the boundary layer. We may also infer that the smaller the value of $H$, the less likely is the boundary layer to separate. ***H* depends on the shape of the velocity profile – hence its name, shape factor**.

***Discussion*** In fact, at the separation point of a laminar boundary layer, $H \approx 3.5$.

---

**10-98**

**Solution** We are to calculate $H$ for an infinitesimally thin boundary layer.

***Analysis*** By definition,

*Shape factor*:
$$H = \frac{\delta^*}{\theta} = \frac{\int_0^\infty \left(1 - \frac{u}{U}\right) dy}{\int_0^\infty \frac{u}{U}\left(1 - \frac{u}{U}\right) dy} \tag{1}$$

But for the limiting case under consideration, $u/U = 1$ through the entire boundary layer, yielding $\delta^* = 0$ and $\theta = 0$. To calculate the ratio in Eq. 1, we use l'Hopital's rule, where the variable $u$ approaches $U$ in the limit,

$$H = \lim_{u \to U} \frac{\frac{d}{du}\int_0^\infty \left(1 - \frac{u}{U}\right) dy}{\frac{d}{du}\int_0^\infty \frac{u}{U}\left(1 - \frac{u}{U}\right) dy} = \lim_{u \to U} \frac{-\int_0^\infty \frac{1}{U} dy}{\int_0^\infty \left(\frac{1}{U} - 2\frac{u}{U}\frac{1}{U}\right) dy} = \frac{\int_0^\infty \frac{1}{U} dy}{\int_0^\infty \frac{1}{U} dy} = \mathbf{1}$$

In other words, $H$ is always greater than unity for any real boundary layer.

***Discussion*** Since turbulent boundary layers are fuller than laminar boundary layers, it is no surprise that $H_{\text{turbulent}}$ is closer to unity than is $H_{\text{laminar}}$.

**10-99**

**Solution** We are to integrate an expression for $\delta$.

***Analysis*** We start with Eq. 5 of Example 10-14,

$$\frac{d\delta}{dx} = \frac{72}{14} 0.027 \left(\text{Re}_x\right)^{-1/7} = 0.139 \left(\frac{Ux}{\nu}\right)^{-1/7}$$

Integration with respect to $x$ yields

$$\delta = \frac{7}{6}(0.139)\left(\frac{Ux}{\nu}\right)^{6/7} \frac{\nu}{U} \quad \to \quad \frac{\delta}{x} = 0.162 \left(\frac{Ux}{\nu}\right)^{6/7} \frac{\nu}{Ux}$$

or, collecting terms, rounding to two digits, and setting $Ux/\nu = \text{Re}_x$,

$$\boxed{\frac{\delta}{x} \approx \frac{0.16}{\left(\text{Re}_x\right)^{1/7}}}$$

***Discussion*** This approximate result is based on the 1/7th power law.

## 10-100

**Solution** We are to generate an expression for $\delta/x$.

***Analysis*** For a flat plate, the Kármán integral equation reduces to

$$C_{f,x} = 2\frac{d\theta}{dx} = 2(0.097)\frac{d\delta}{dx}$$

where we have used the given expression for $\theta$ as a function of $\delta$. Substitution of the given expression for $C_{f,x}$ gives

$$\frac{d\delta}{dx} = \frac{C_{f,x}}{2(0.097)} = \frac{0.059(\text{Re}_x)^{-1/5}}{0.194} = 0.304\left(\frac{Ux}{\nu}\right)^{-1/5}$$

Integration with respect to $x$ yields

$$\delta = \frac{5}{4}(0.304)\left(\frac{Ux}{\nu}\right)^{4/5}\frac{\nu}{U} \quad \rightarrow \quad \frac{\delta}{x} = 0.380\left(\frac{Ux}{\nu}\right)^{4/5}\frac{\nu}{Ux}$$

or, collecting terms, rounding to two digits, and setting $Ux/\nu = \text{Re}_x$,

$$\boxed{\frac{\delta}{x} \approx \frac{0.38}{(\text{Re}_x)^{1/5}}}$$

**The result is identical to that of Table 10-4, column (b)**.

***Discussion*** The Kármán integral equation is useful for obtaining approximate relations like those of Table 10-4.

---

## Review Problems

---

## 10-101C

**Solution**

(*a*) ***True***: We do not have to make the 2-D approximation in order to define the velocity potential function – $\phi$ can be defined for any flow if the vorticity is zero.

(*b*) ***False***: The stream function is definable for any two-dimensional flow field, regardless of the value of vorticity.

(*c*) ***True***: The velocity potential function is valid only for irrotational flow regions where the vorticity is zero.

(d) ***True***: The stream function is defined from the continuity equation, and is valid only for two-dimensional flows. Note that some researchers have defined three-dimensional forms of the stream function, but these are beyond the scope of the present introductory text book.

---

**10-102**

**Solution** We are to compute the viscous term for the given velocity field, and show that the flow is rotational.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis*** We consider the viscous terms of the $\theta$ component of the Navier-Stokes equation,

*Viscous terms*:
$$\mu\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_\theta}{\partial r}\right)-\frac{u_\theta}{r^2}+\frac{1}{r^2}\frac{\partial^2 u_\theta}{\partial\theta^2}-\frac{2}{r^2}\frac{\partial u_r}{\partial\theta}+\frac{\partial^2 u_\theta}{\partial z^2}\right)$$
$$=\mu\left(\frac{\omega}{r}\quad-\frac{\omega}{r}\quad+0\quad-0\quad+0\right)=0$$

The viscous terms are zero, implying that **there are no net viscous forces acting on fluid elements**. This does not necessarily mean that the flow is inviscid – it could also mean that the flow is irrotational, since the viscous terms disappear in both inviscid and irrotational flows. We obtain the $z$ component of vorticity in cylindrical coordinates from Chap. 4,

*z component of vorticity*:
$$\zeta_z=\frac{1}{r}\left(\frac{\partial(ru_\theta)}{\partial r}-\frac{\partial u_r}{\partial\theta}\right)=\frac{1}{r}\left(\frac{\partial(\omega r^2)}{\partial r}-0\right)=2\omega$$

Thus, since the vorticity is non-zero, **this flow field is rotational**. Finally, since the flow is not irrotational, the only other way that the net viscous force can be zero is if the flow is inviscid. We conclude, then, that **this flow field is also inviscid**.

***Discussion*** The vorticity is twice the angular velocity, as discussed in Chap. 4.

---

**10-103**

**Solution** We are to calculate the viscous stress tensor for a given velocity field.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis*** The viscous stress tensor is given in Chap. 9 as

*Viscous stress tensor in cylindrical coordinates*:
$$\tau_{ij}=\begin{pmatrix}\tau_{rr} & \tau_{r\theta} & \tau_{rz}\\ \tau_{\theta r} & \tau_{\theta\theta} & \tau_{\theta z}\\ \tau_{zr} & \tau_{z\theta} & \tau_{zz}\end{pmatrix}=\begin{pmatrix}2\mu\frac{\partial u_r}{\partial r} & \mu\left(r\frac{\partial}{\partial r}\left(\frac{u_\theta}{r}\right)+\frac{1}{r}\frac{\partial u_r}{\partial\theta}\right) & \mu\left(\frac{\partial u_r}{\partial z}+\frac{\partial u_z}{\partial r}\right)\\ \mu\left(r\frac{\partial}{\partial r}\left(\frac{u_\theta}{r}\right)+\frac{1}{r}\frac{\partial u_r}{\partial\theta}\right) & 2\mu\left(\frac{1}{r}\frac{\partial u_\theta}{\partial\theta}+\frac{u_r}{r}\right) & \mu\left(\frac{\partial u_\theta}{\partial z}+\frac{1}{r}\frac{\partial u_z}{\partial\theta}\right)\\ \mu\left(\frac{\partial u_r}{\partial z}+\frac{\partial u_z}{\partial r}\right) & \mu\left(\frac{\partial u_\theta}{\partial z}+\frac{1}{r}\frac{\partial u_z}{\partial\theta}\right) & 2\mu\frac{\partial u_z}{\partial z}\end{pmatrix}\quad(1)$$

We plug in the velocity field from Problem 10-102 into Eq. 1, and we get

$$\tau_{ij}=\begin{pmatrix}\tau_{rr} & \tau_{r\theta} & \tau_{rz}\\ \tau_{\theta r} & \tau_{\theta\theta} & \tau_{\theta z}\\ \tau_{zr} & \tau_{z\theta} & \tau_{zz}\end{pmatrix}=\begin{pmatrix}0&0&0\\0&0&0\\0&0&0\end{pmatrix}\quad(2)$$

Thus, we conclude that there are no viscous stresses in this flow field (solid body rotation). Thus, **this flow can be considered inviscid**.

***Discussion*** Since the fluid moves as a solid body, no fluid particles move relative to any other fluid particles; hence we expect no viscous stresses.

---

## 10-104

**Solution** We are to compute the viscous term for the velocity field and show that the flow is irrotational.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis*** First, we write out and simplify the viscous terms of the $\theta$ component of the Navier-Stokes equation,

$$\mu\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial u_\theta}{\partial r}\right)-\frac{u_\theta}{r^2}+\frac{1}{r^2}\frac{\partial^2 u_\theta}{\partial \theta^2}-\frac{2}{r^2}\frac{\partial u_r}{\partial \theta}+\frac{\partial^2 u_\theta}{\partial z^2}\right)=\mu\left(\frac{\Gamma}{2\pi r^3}-\frac{\Gamma}{2\pi r^3}\quad+0\quad-0\quad+0\right)=0$$

The viscous terms are zero, implying that **there are no net viscous forces acting on fluid elements**. This does not necessarily mean that the flow is inviscid – it could also mean that the flow is irrotational, since the viscous terms disappear in both inviscid and irrotational flows.

We obtain the $z$ component of vorticity in cylindrical coordinates from Chap. 4,

*z component of vorticity*:
$$\zeta_z=\frac{1}{r}\left(\frac{\partial\left(ru_\theta\right)}{\partial r}-\frac{\partial u_r}{\partial \theta}\right)=\frac{1}{r}\left(0-0\right)=0$$

Thus, since the vorticity is zero, **this flow field is irrotational**.

***Discussion*** We cannot say for sure whether the flow is inviscid unless we calculate the viscous shear stresses, as in the following problem.

---

## 10-105

**Solution** We are to calculate the viscous stress tensor for a given velocity field.

***Assumptions*** **1** The flow is steady. **2** The flow is two-dimensional in the $r$-$\theta$ plane.

***Analysis*** The viscous stress tensor in cylindrical coordinates is given in Chap. 9 as

$$\tau_{ij}=\begin{pmatrix}\tau_{rr} & \tau_{r\theta} & \tau_{rz}\\ \tau_{\theta r} & \tau_{\theta\theta} & \tau_{\theta z}\\ \tau_{zr} & \tau_{z\theta} & \tau_{zz}\end{pmatrix}=\begin{pmatrix}2\mu\frac{\partial u_r}{\partial r} & \mu\left(r\frac{\partial}{\partial r}\left(\frac{u_\theta}{r}\right)+\frac{1}{r}\frac{\partial u_r}{\partial \theta}\right) & \mu\left(\frac{\partial u_r}{\partial z}+\frac{\partial u_z}{\partial r}\right)\\ \mu\left(r\frac{\partial}{\partial r}\left(\frac{u_\theta}{r}\right)+\frac{1}{r}\frac{\partial u_r}{\partial \theta}\right) & 2\mu\left(\frac{1}{r}\frac{\partial u_\theta}{\partial \theta}+\frac{u_r}{r}\right) & \mu\left(\frac{\partial u_\theta}{\partial z}+\frac{1}{r}\frac{\partial u_z}{\partial \theta}\right)\\ \mu\left(\frac{\partial u_r}{\partial z}+\frac{\partial u_z}{\partial r}\right) & \mu\left(\frac{\partial u_\theta}{\partial z}+\frac{1}{r}\frac{\partial u_z}{\partial \theta}\right) & 2\mu\frac{\partial u_z}{\partial z}\end{pmatrix}\qquad(1)$$

We plug in the velocity field from Problem 10-104 into Eq. 1, and we get

$$\tau_{ij}=\begin{pmatrix}\tau_{rr} & \tau_{r\theta} & \tau_{rz}\\ \tau_{\theta r} & \tau_{\theta\theta} & \tau_{\theta z}\\ \tau_{zr} & \tau_{z\theta} & \tau_{zz}\end{pmatrix}=\begin{pmatrix}0 & \mu\frac{\Gamma}{\pi r^2} & 0\\ \mu\frac{\Gamma}{\pi r^2} & 0 & 0\\ 0 & 0 & 0\end{pmatrix}\qquad(2)$$

Thus, we conclude that there are indeed some non-zero viscous stresses in this flow field. Hence, **this flow is not inviscid, even though it is irrotational**.

***Discussion*** The fluid particles move relative to each other, generating viscous shear stresses. However, the *net* viscous force on a fluid particle is zero since the flow is irrotational.

---

## 10-106

**Solution** We are to calculate modified pressure $P'$ and sketch profiles of $P'$ at two vertical locations in the pipe.

***Assumptions*** **1** The flow is incompressible. **2** Gravity acts vertically downward. **3** There are no free surface effects in this flow field.

***Analysis*** By definition, modified pressure $P' = P + \rho gz$. So we add hydrostatic pressure component $\rho gz$ to the given profile $P = P_{atm}$ to obtain the profile for $P'$. At any $z$ location in the pipe,

*Modified pressure*: $P' = P + \rho gz \quad \rightarrow \quad P' = P_{atm} + \rho gz$

We see that $P'$ is uniform at any vertical location ($P'$ does not vary radially), but $P'$ varies with elevation $z$. At $z = z_1$,

*Modified pressure at $z_1$*: $\boxed{P_1' = P_{atm} + \rho g z_1}$ (1)

and at $z = z_2$,

*Modified pressure at $z_2$*: $\boxed{P_2' = P_{atm} + \rho g z_2}$ (2)

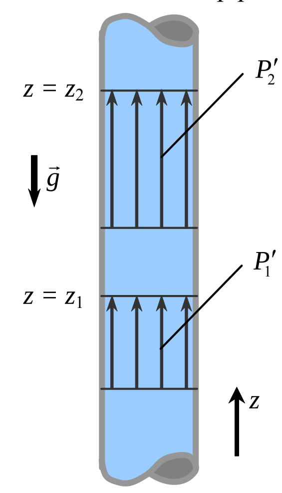



**FIGURE 1**
Modified pressure profiles at two vertical locations in the pipe.

Comparing Eqs. 1 and 2, the modified pressure is higher at location $z_2$ since $z_2$ is higher in elevation than $z_1$. In this problem there is no forced pressure gradient in terms of actual pressure. However, in terms of *modified* pressure, there is a linearly decreasing modified pressure along the axis of the pipe. In other words, there *is* a forced pressure gradient in terms of modified pressure.

***Discussion*** Since modified pressure eliminates the gravity term from the Navier-Stokes equation, we have replaced the effect of gravity by a gradient of modified pressure $P'$.

---

## 10-107

**Solution** We are to calculate the required pressure drop between two axial locations of a horizontal pipe that would yield the same volume flow rate as that of the vertical pipe of Problem 10-106.

***Assumptions*** **1** The flow is incompressible. **2** Gravity acts vertically downward. **3** There are no free surface effects in this flow field.

***Analysis*** For the vertical case of Problem 10-106, we know that at $z = z_1$,

*Modified pressure at $z_1$*: $P_1' = P_{atm} + \rho g z_1$ (1)

and at $z = z_2$,

*Modified pressure at $z_2$*: $P_2' = P_{atm} + \rho g z_2$ (2)

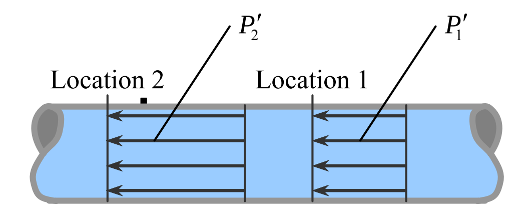



**FIGURE 1**
Modified pressure profiles at two horizontal locations in the pipe.

Since modified pressure effectively eliminates gravity from the problem, we expect that at the same volume flow rate, *the difference in modified pressure from $z_2$ to $z_1$ does not change with changes in the orientation of the pipe*. From Eqs. 1 and 2,

*Change in modified pressure from $z_2$ to $z_1$*: $P_2' - P_1' = \rho g (z_2 - z_1)$ (3)

The modified pressure profiles at two axial locations in the horizontal pipe are sketched in Fig. 1. We convert the modified pressures in Eq. 3 to actual pressures using the definition of modified pressure, $P = P' - \rho gz$. We note however, that for the horizontal pipe, $z$ does not change along the pipe. Thus we conclude that the required difference in actual pressure is

*Change in pressure from location 2 to location 1*: $\boxed{P_2 - P_1 = \rho g (z_2 - z_1)}$ (4)

where $z_2$ and $z_1$ are the elevations of the vertical pipe case.

***Discussion*** In order to achieve the same flow rate, the forced pressure gradient in the horizontal case must be the same as the hydrostatic pressure difference supplied by gravity in the vertical case.

---

## Design and Essay Problem

### 10-108

**Solution** We are to discuss the velocity overshoot in Fig. 10-136.

***Analysis*** **The velocity overshoot is a direct result of the displacement effect and the effect of inertia**. At very *low* values of $Re_L$ (less than about $10^1$), where the displacement effect is most prominent, the velocity overshoot is almost non-existent. This can be explained by the lack of inertia at these low Reynolds numbers. Without inertia, there is no mechanism to accelerate the flow around the plate; rather, viscosity *retards* the flow everywhere in the vicinity of the plate, and the influence of the plate extends tens of plate lengths beyond the plate in all directions. At *moderate* values of Reynolds number ($Re_L$ between about $10^1$ and $10^4$), the displacement effect is significant, and inertial terms are no longer negligible. Hence, fluid is able to accelerate around the plate and the velocity overshoot is significant. At very *high* values of Reynolds number ($Re_L > 10^4$), inertial terms dominate viscous terms, and the boundary layer is so thin that the displacement effect is almost negligible – the small displacement effect leads to very small velocity overshoot at high Reynolds numbers.

***Discussion*** We can imagine that the flat plate appears thicker from the point of view of the outer flow, and therefore, the flow must accelerate around this "fat" plate.



Solutions Manual for

Fluid Mechanics: Fundamentals and Applications

by Çengel & Cimbala

---

# CHAPTER 11
# FLOW OVER BODIES: DRAG AND LIFT

---

## Drag, Lift, and Drag Coefficients of Common Geometries

### 11-1C

**Solution** We are to explain when a flow is 2-D, 3-D, and axisymmetric.

***Analysis*** The flow over a body is said to be *two-dimensional* when **the body is very long and of constant cross-section, and the flow is normal to the body** (such as the wind blowing over a long pipe perpendicular to its axis). There is no significant flow along the axis of the body. The flow along a body that **possesses symmetry along an axis in the flow direction** is said to be *axisymmetric* (such as a bullet piercing through air). Flow over a body that **cannot be modeled as two-dimensional or axisymmetric** is *three-dimensional*. The flow over a car is three-dimensional.

***Discussion*** As you might expect, 3-D flows are much more difficult to analyze than 2-D or axisymmetric flows.

### 11-2C

**Solution** We are to discuss the difference between upstream and free-stream velocity.

***Analysis*** The **velocity of the fluid relative to the immersed solid body sufficiently far away from a body** is called the *free-stream velocity*, *V*. The *upstream* (or *approach*) *velocity V* is the **velocity of the approaching fluid far ahead of the body**. These two velocities are equal if the flow is uniform and the body is small relative to the scale of the free-stream flow.

***Discussion*** This is a subtle difference, and the two terms are often used interchangeably.

### 11-3C

**Solution** We are to discuss the difference between streamlined and blunt bodies.

***Analysis*** A body is said to be *streamlined* if a conscious effort is made to **align its shape with the anticipated streamlines in the flow**. Otherwise, a body tends to **block the flow**, and is said to be *blunt*. A tennis ball is a blunt body (unless the velocity is very low and we have "creeping flow").

***Discussion*** In creeping flow, the streamlines align themselves with the shape of any body – this is a much different regime than our normal experiences with flows in air and water. A low-drag body shape in creeping flow looks much different than a low-drag shape in high Reynolds number flow.

### 11-4C

**Solution** We are to discuss applications in which a large drag is desired.

***Analysis*** Some applications in which a large drag is desirable: **parachuting**, **sailing**, and the **transport of pollens**.

***Discussion*** When sailing efficiently, however, the *lift* force on the sail is more important than the drag force in propelling the boat.

### 11-5C

**Solution** We are to define drag and discuss why we usually try to minimize it.

***Analysis*** The **force a flowing fluid exerts on a body in the flow direction** is called *drag*. **Drag is caused by friction between the fluid and the solid surface, and the pressure difference between the front and back of the body**. We try to minimize drag in order to **reduce fuel consumption** in vehicles, **improve safety and durability** of structures subjected to high winds, and to **reduce noise and vibration**.

***Discussion*** In some applications, such as parachuting, high drag rather than low drag is desired.

**11-6C**

**Solution** We are to define lift, and discuss its cause and the contribution of wall shear to lift.

***Analysis*** The **force a flowing fluid exerts on a body in the normal direction to flow that tends to move the body in that direction** is called *lift*. It is caused by the **components of the pressure and wall shear forces in the direction normal to the flow**. The wall shear contributes to lift (unless the body is very slim), but **its contribution is usually small**.

***Discussion*** Typically the nonsymmetrical *shape* of the body is what causes the lift force to be produced.

---

**11-7C**

**Solution** We are to explain how to calculate the drag coefficient, and discuss the appropriate area.

***Analysis*** When the drag force $F_D$, the upstream velocity $V$, and the fluid density $\rho$ are measured during flow over a body, the drag coefficient is determined from

$$\boxed{C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A}}$$

where ***A*** **is ordinarily the *frontal area*** (the area projected on a plane normal to the direction of flow) of the body.

***Discussion*** In some cases, however, such as flat plates aligned with the flow or airplane wings, the planform area is used instead of the frontal area. Planform area is the area projected on a plane parallel to the direction of flow and normal to the lift force.

---

**11-8C**

**Solution** We are to explain how to calculate the lift coefficient, and discuss the appropriate area.

***Analysis*** When the lift force $F_L$, the upstream velocity $V$, and the fluid density $\rho$ are measured during flow over a body, the lift coefficient can be determined from

$$\boxed{C_L = \frac{F_L}{\frac{1}{2}\rho V^2 A}}$$

where ***A*** **is ordinarily the *planform area***, which is the area that would be seen by a person looking at the body from above in a direction normal to the body.

***Discussion*** In some cases, however, such as flat plates aligned with the flow or airplane wings, the planform area is used instead of the frontal area. Planform area is the area projected on a plane parallel to the direction of flow and normal to the lift force.

---

**11-9C**

**Solution** We are to define the frontal area of a body and discuss its applications.

***Analysis*** The *frontal area* of a body is the **area seen by a person when looking from upstream** (the area projected on a plane normal to the direction of flow of the body). The frontal area is appropriate to use in drag and lift calculations for blunt bodies such as cars, cylinders, and spheres.

***Discussion*** The drag force on a body is proportional to both the drag coefficient *and* the frontal area. Thus, one is able to reduce drag by reducing the drag coefficient or the frontal area (or both).

---

**11-10C**

**Solution** We are to define the planform area of a body and discuss its applications.

***Analysis*** The *planform area* of a body is the **area that would be seen by a person looking at the body from above in a direction normal to flow**. The planform area is the area projected on a plane parallel to the direction of flow and normal to the lift force. The planform area is appropriate to use in drag and lift calculations for **slender bodies such as flat plate and airfoils when the frontal area is very small**.

***Discussion*** Consider for example an extremely thin flat plate aligned with the flow. The frontal area is nearly zero, and is therefore not appropriate to use for calculation of drag or lift coefficient.

---

**11-11C**

**Solution** We are to define and discuss terminal velocity.

***Analysis*** The **maximum velocity a free falling body can attain** is called the *terminal velocity*. **It is determined by setting the weight of the body equal to the drag and buoyancy forces**, $W = F_D + F_B$.

***Discussion*** When discussing the settling of small dust particles, terminal velocity is also called *terminal settling speed* or *settling velocity*.

---

**11-12C**

**Solution** We are to discuss the difference between skin friction drag and pressure drag, and which is more significant for slender bodies.

***Analysis*** The **part of drag that is due directly to wall shear stress $\tau_w$** is called the *skin friction drag* $F_{D,\text{ friction}}$ since it is caused by frictional effects, and **the part that is due directly to pressure *P* and depends strongly on the shape of the body** is called the *pressure drag* $F_{D,\text{ pressure}}$. For slender bodies such as airfoils, **the friction drag is usually more significant**.

***Discussion*** For blunt bodies, on the other hand, pressure drag is usually more significant than skin friction drag.

---

**11-13C**

**Solution** We are to discuss the effect of surface roughness on drag coefficient.

***Analysis*** The friction drag coefficient is **independent of surface roughness** in *laminar flow*, but is a **strong function of surface roughness** in *turbulent flow* due to surface roughness elements protruding farther into the viscous sublayer.

***Discussion*** If the roughness is very large, however, the drag on bodies is increased even for laminar flow, due to pressure effects on the roughness elements.

---

**11-14C**

**Solution** We are to discuss how drag coefficient varies with Reynolds number.

***Analysis*** (*a*) In general, the drag coefficient **decreases with Reynolds number at low and moderate Reynolds numbers**. (*b*) The drag coefficient is **nearly independent of Reynolds number at high Reynolds numbers** ($Re > 10^4$).

***Discussion*** When the drag coefficient is independent of Re at high values of Re, we call this Reynolds number independence.

---

**11-15C**

**Solution** We are to discuss the effect of adding a fairing to a circular cylinder.

***Analysis*** As a result of attaching fairings to the front and back of a cylindrical body at high Reynolds numbers, (*a*) **friction drag increases**, (*b*) **pressure drag decreases**, and (*c*) **total drag decreases**.

***Discussion*** In creeping flow (very low Reynolds numbers), however, adding a fairing like this would actually *increase* the overall drag, since the surface area and therefore the skin friction drag would increase significantly.

---

**11-16C**

**Solution** We are to discuss the effect of streamlining, and its effect on friction drag and pressure drag.

***Analysis*** As a result of streamlining, (*a*) **friction drag increases**, (*b*) **pressure drag decreases**, and (*c*) **total drag decreases** at high Reynolds numbers (the general case), but increases at very low Reynolds numbers (creeping flow) since the friction drag dominates at low Reynolds numbers.

***Discussion*** Streamlining can significantly reduce the overall drag on a body at high Reynolds number.

---

**11-17C**

**Solution** We are to define and discuss flow separation.

***Analysis*** At sufficiently high velocities, **the fluid stream detaches itself from the surface of the body**. This is called *separation*. It is caused by a fluid flowing over a curved surface at a high velocity (or technically, by adverse pressure gradient). Separation increases the drag coefficient drastically.

***Discussion*** A boundary layer has a hard time resisting an adverse pressure gradient, and is likely to separate. A turbulent boundary layer is in general more resilient to flow separation than a laminar flow.

---

**11-18C**

**Solution** We are to define and discuss drafting.

***Analysis*** *Drafting* is when **a moving body follows another moving body by staying close behind in order to reduce drag**. It reduces the pressure drag and thus the drag coefficient for the drafted body by **taking advantage of the low pressure wake region of the moving body in front**.

***Discussion*** We often see drafting in automobile and bicycle racing.

---

**11-19C**

**Solution** We are to compare the fuel efficiency of two cars.

***Analysis*** The car that is contoured to resemble an ellipse has a smaller drag coefficient and thus smaller air resistance, and thus **the elliptical car is more likely to be more fuel efficient than a car with sharp corners**.

***Discussion*** However, sharp corners promote a fixed flow separation location, whereas rounded corners can lead to unsteadiness in the flow. For this reason, rounding is usually, but not *always* the best solution.

---

**11-20C**

**Solution** We are to compare the speed of two bicyclists.

***Analysis*** **The bicyclist who leans down and brings his body closer to his knees goes faster** since the frontal area and thus the drag force is less in that position. The drag coefficient also goes down somewhat, but this is a secondary effect.

***Discussion*** This is easily experienced when riding a bicycle down a long hill.

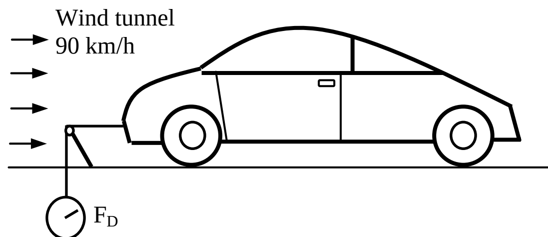
---

## 11-21

**Solution** The drag force acting on a car is measured in a wind tunnel. The drag coefficient of the car at the test conditions is to be determined.

***Assumptions*** **1** The flow of air is steady and incompressible. **2** The cross-section of the tunnel is large enough to simulate free flow over the car. **3** The bottom of the tunnel is also moving at the speed of air to approximate actual driving conditions or this effect is negligible. **4** Air is an ideal gas.

***Properties*** The density of air at 1 atm and 25°C is $\rho$ = 1.164 kg/m$^3$.




***Analysis*** The drag force acting on a body and the drag coefficient are given by

$$F_D = C_D A \frac{\rho V^2}{2} \quad \text{and} \quad C_D = \frac{2F_D}{\rho A V^2}$$

where $A$ is the frontal area. Substituting and noting that 1 m/s = 3.6 km/h, the drag coefficient of the car is determined to be

$$C_D = \frac{2 \times (350\text{ N})}{(1.164\text{ kg/m}^3)(1.40 \times 1.65\text{ m}^2)(90/3.6\text{ m/s})^2}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right) = \mathbf{0.42}$$

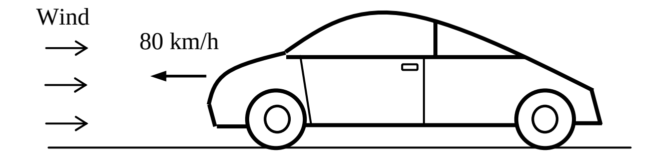
***Discussion*** Note that the drag coefficient depends on the design conditions, and its value will be different at different conditions. Therefore, the published drag coefficients of different vehicles can be compared meaningfully only if they are determined under identical conditions. This shows the importance of developing standard testing procedures in industry.

---

## 11-22

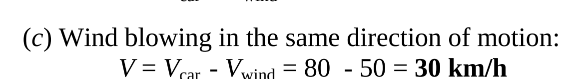
**Solution** A car is moving at a constant velocity. The upstream velocity to be used in fluid flow analysis is to be determined for the cases of calm air, wind blowing against the direction of motion of the car, and wind blowing in the same direction of motion of the car.

***Analysis*** In fluid flow analysis, the velocity used is the relative velocity between the fluid and the solid body. Therefore:




(*a*) Calm air: $V = V_{car}$ = **80 km/h**

(*b*) Wind blowing against the direction of motion:
$V = V_{car} + V_{wind}$ = 80 + 30 = **110 km/h**



(*c*) Wind blowing in the same direction of motion:
$V = V_{car} - V_{wind}$ = 80 - 50 = **30 km/h**

***Discussion*** Note that the wind and car velocities are added when they are in opposite directions, and subtracted when they are in the same direction.

---

## 11-23

**Solution** The resultant of the pressure and wall shear forces acting on a body is given. The drag and the lift forces acting on the body are to be determined.

***Analysis*** The drag and lift forces are determined by decomposing the resultant force into its components in the flow direction and the normal direction to flow,

V
$F_R$=700 N
35°

*Drag force*: $F_D = F_R \cos\theta = (700\text{ N})\cos 35° = \mathbf{573\ N}$

*Lift force*: $F_L = F_R \sin\theta = (700\text{ N})\sin 35° = \mathbf{402\ N}$

***Discussion*** Note that the greater the angle between the resultant force and the flow direction, the greater the lift.

---

**11-24**

**Solution** The total drag force acting on a spherical body is measured, and the pressure drag acting on the body is calculated by integrating the pressure distribution. The friction drag coefficient is to be determined.

***Assumptions*** **1** The flow of air is steady and incompressible. **2** The surface of the sphere is smooth. **3** The flow over the sphere is turbulent (to be verified).

***Properties*** The density and kinematic viscosity of air at 1 atm and 5°C are $\rho$ = 1.269 kg/m$^3$ and $\nu$ = 1.382×10$^{-5}$ m$^2$/s. The drag coefficient of sphere in turbulent flow is $C_D$ = 0.2, and its frontal area is $A = \pi D^2/4$ (Table 11-2).

***Analysis*** The total drag force is the sum of the friction and pressure drag forces. Therefore,

$$F_{D,\text{friction}} = F_D - F_{D,\text{pressure}} = 5.2 - 4.9 = 0.3\ \text{N}$$

where $F_D = C_D A \dfrac{\rho V^2}{2}$ and $F_{D,\text{friction}} = C_{D,\text{friction}} A \dfrac{\rho V^2}{2}$

Air
$V$
$D$ = 12 cm

Taking the ratio of the two relations above gives

$$C_{D,\text{friction}} = \frac{F_{D,\text{friction}}}{F_D} C_D = \frac{0.3\ \text{N}}{5.2\ \text{N}}(0.2) = \mathbf{0.0115}$$

Now we need to verify that the flow is turbulent. This is done by calculating the flow velocity from the drag force relation, and then the Reynolds number:

$$F_D = C_D A \frac{\rho V^2}{2} \rightarrow V = \sqrt{\frac{2F_D}{\rho C_D A}} = \sqrt{\frac{2(5.2\ \text{N})}{(1.269\ \text{kg/m}^3)(0.2)[\pi(0.12\ \text{m})^2/4]}\left(\frac{1\ \text{kg}\cdot\text{m/s}^2}{1\ \text{N}}\right)} = 60.2\ \text{m/s}$$

$$\text{Re} = \frac{VD}{\nu} = \frac{(60.2\ \text{m/s})(0.12\ \text{m})}{1.382\times10^{-5}\ \text{m}^2/s} = 5.23\times10^5$$

which is greater than 2×10$^5$. Therefore, the flow is turbulent as assumed.

***Discussion*** Note that knowing the flow regime is important in the solution of this problem since the total drag coefficient for a sphere is 0.5 in laminar flow and 0.2 in turbulent flow.

**11-25E**

**Solution** The frontal area of a car is reduced by redesigning. The amount of fuel and money saved per year as a result are to be determined.



***Assumptions*** **1** The car is driven 12,000 miles a year at an average speed of 55 km/h. **2** The effect of reduction of the frontal area on the drag coefficient is negligible.

***Properties*** The densities of air and gasoline are given to be 0.075 lbm/ft$^3$ and 50 lbm/ft$^3$, respectively. The heating value of gasoline is given to be 20,000 Btu/lbm. The drag coefficient is $C_D$ = 0.3 for a passenger car (Table 11-2).

***Analysis*** The drag force acting on a body is determined from

$$F_D = C_D A \frac{\rho V^2}{2}$$

where $A$ is the frontal area of the body. The drag force acting on the car before redesigning is

$$F_D = 0.3(18\text{ ft}^2)\frac{(0.075\text{ lbm/ft}^3)(55\text{ mph})^2}{2}\left(\frac{1.4667\text{ ft/s}}{1\text{ mph}}\right)^2\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = 40.9\text{ lbf}$$

Noting that work is force times distance, the amount of work done to overcome this drag force and the required energy input for a distance of 12,000 miles are

$$W_{\text{drag}} = F_D L = (40.9\text{ lbf})(12{,}000\text{ miles/year})\left(\frac{5280\text{ ft}}{1\text{ mile}}\right)\left(\frac{1\text{ Btu}}{778.169\text{ lbf}\cdot\text{ft}}\right) = 3.33\times10^6\text{ Btu/year}$$

$$E_{in} = \frac{W_{\text{drag}}}{\eta_{\text{car}}} = \frac{3.33\times10^6\text{ Btu/year}}{0.32} = 1.041\times10^7\text{ Btu/year}$$

Then the amount and costs of the fuel that supplies this much energy are

$$\text{Amont of fuel} = \frac{m_{\text{fuel}}}{\rho_{\text{fuel}}} = \frac{E_{\text{in}}/\text{HV}}{\rho_{\text{fuel}}} = \frac{(1.041\times10^7\text{ Btu/year})/(20{,}000\text{ Btu/lbm})}{50\text{ lbm/ft}^3} = 10.41\text{ ft}^3/\text{year}$$

$$\text{Cost} = (\text{Amount of fuel})(\text{Unit cost}) = (10.41\text{ ft}^3/\text{year})(\$2.20/\text{gal})\left(\frac{7.4804\text{ gal}}{1\text{ ft}^3}\right) = \$171.3/\text{year}$$

That is, the car uses 10.41 ft$^3$ = 77.9 gallons of gasoline at a cost of $171.3 per year to overcome the drag.

The drag force and the work done to overcome it are directly proportional to the frontal area. Then the percent reduction in the fuel consumption due to reducing frontal area is equal to the percent reduction in the frontal area:

$$\text{Reduction ratio} = \frac{A - A_{\text{new}}}{A} = \frac{18-15}{18} = 0.167$$

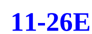
$$\text{Amount reduction} = (\text{Reduction ratio})(\text{Amount})$$

$$= 0.167(77.9\text{ gal/year}) = \mathbf{13.0\ gal/year}$$

$$\text{Cost reduction} = (\text{Reduction ratio})(\text{Cost}) = 0.167(\$171.3/\text{year}) = \mathbf{\$28.6/year}$$

Therefore, reducing the frontal area reduces the fuel consumption due to drag by 16.7%.

***Discussion*** Note from this example that significant reductions in drag and fuel consumption can be achieved by reducing the frontal area of a vehicle.

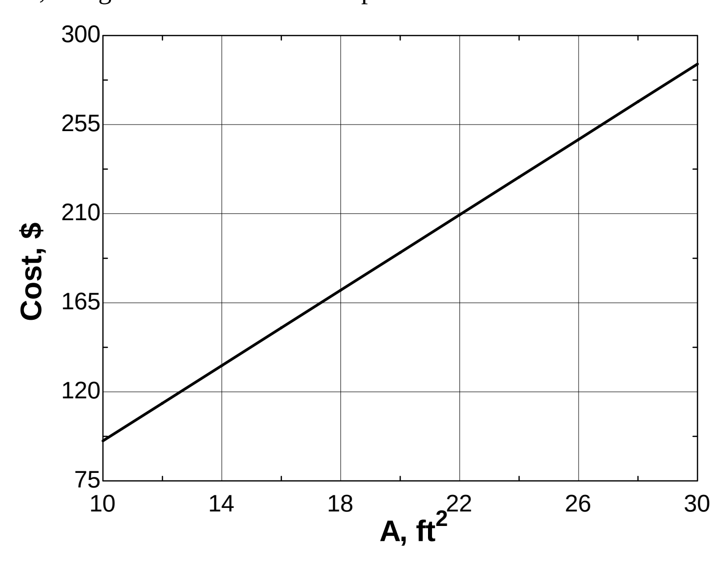


**11-26E** 

**Solution** The previous problem is reconsidered. The effect of frontal area on the annual fuel consumption of the car as the frontal area varied from 10 to 30 ft$^2$ in increments of 2 ft$^2$ are to be investigated.

***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
CD=0.3
rho=0.075 "lbm/ft3"
V=55*1.4667 "ft/s"
Eff=0.32
Price=2.20 "$/gal"
efuel=20000 "Btu/lbm"
rho_gas=50 "lbm/ft3"
L=12000*5280 "ft"
FD=CD*A*(rho*V^2)/2/32.2 "lbf"
Wdrag=FD*L/778.169 "Btu"
Ein=Wdrag/Eff
m=Ein/efuel "lbm"
Vol=(m/rho_gas)*7.4804 "gal"
Cost=Vol*Price
```




| $A$, ft$^2$ | $F_{drag}$ lbf | Amount, gal | Cost, $ |
|---|---|---|---|
| 10 | 22.74 | 43.27 | 95.2 |
| 12 | 27.28 | 51.93 | 114.2 |
| 14 | 31.83 | 60.58 | 133.3 |
| 16 | 36.38 | 69.24 | 152.3 |
| 18 | 40.92 | 77.89 | 171.4 |
| 20 | 45.47 | 86.55 | 190.4 |
| 22 | 50.02 | 95.2 | 209.4 |
| 24 | 54.57 | 103.9 | 228.5 |
| 26 | 59.11 | 112.5 | 247.5 |
| 28 | 63.66 | 121.2 | 266.6 |
| 30 | 68.21 | 129.8 | 285.6 |

***Discussion*** As you might expect, the cost goes up linearly with area, since drag force goes up linearly with area.

**11-27** **[*Also solved using EES on enclosed DVD*]**

**Solution** A circular sign is subjected to high winds. The drag force acting on the sign and the bending moment at the bottom of its pole are to be determined.

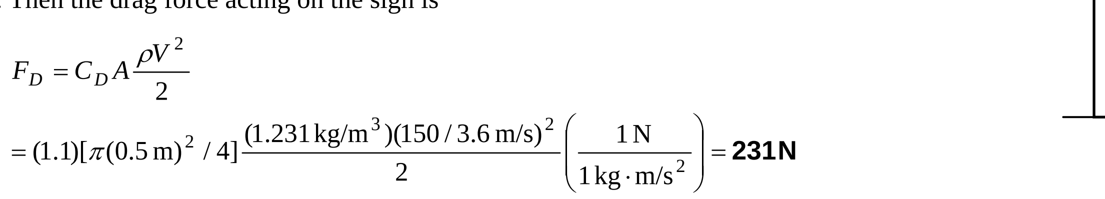
***Assumptions*** **1** The flow of air is steady and incompressible. **2** The drag force on the pole is negligible. **3** The flow is turbulent so that the tabulated value of the drag coefficient can be used.

***Properties*** The drag coefficient for a thin circular disk is $C_D$ = 1.1 (Table 11-2). The density of air at 100 kPa and 10°C = 283 K is

SIGN
50 cm
150 km/h
1.5 m

$$\rho = \frac{P}{RT} = \frac{100\ \text{kPa}}{(0.287\ \text{kPa}\cdot\text{m}^3/\text{kg.K})(283\ \text{K})} = 1.231\ \text{kg/m}^3$$

***Analysis*** The frontal area of a circular plate subjected to normal flow is $A = \pi D^2/4$. Then the drag force acting on the sign is



$$F_D = C_D A \frac{\rho V^2}{2}$$

$$= (1.1)[\pi(0.5\ \text{m})^2/4]\frac{(1.231\ \text{kg/m}^3)(150/3.6\ \text{m/s})^2}{2}\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = \mathbf{231\ N}$$

Noting that the resultant force passes through the center of the stop sign, the bending moment at the bottom of the pole becomes

$$M_{bottom} = F_D \times L = (231\ \text{N})(1.5 + 0.25)\ \text{m} = \mathbf{404\ Nm}$$

***Discussion*** Note that the drag force is equivalent to the weight of over 23 kg of mass. Therefore, the pole must be strong enough to withstand the weight of 23 kg hanged at one of its end when it is held from the other end horizontally.

---

**11-28E**

**Solution** A rectangular billboard is subjected to high winds. The drag force acting on the billboard is to be determined.

***Assumptions*** **1** The flow of air is steady and incompressible. **2** The drag force on the supporting poles are negligible. **3** The flow is turbulent so that the tabulated value of the drag coefficient can be used.

20 ft
8 ft
90 mph
BILLBOARD

***Properties*** The drag coefficient for a thin rectangular plate for normal flow is $C_D$ = 2.0 (Table 11-2). The density of air at 14.3 psia and 40°F = 500 R is

$$\rho = \frac{P}{RT} = \frac{14.3\ \text{psia}}{(0.3704\ \text{psia}\cdot\text{ft}^3/\text{lbm.R})(500\ \text{R})} = 0.0772\ \text{lbm/ft}^3$$

***Analysis*** The drag force acting on the billboard is determined from

$$F_D = C_D A \frac{\rho V^2}{2} = (2.0)(8\times 20\ \text{ft}^2)\frac{(0.0772\ \text{lbm/ft}^3)(90\times 1.4667\ \text{ft/s})^2}{2}\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right) = 6684\ \text{lbf} \cong \mathbf{6680\ lbf}$$

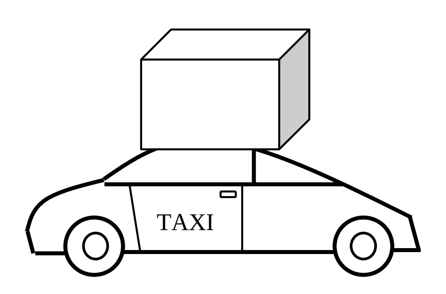
***Discussion*** Note that the drag force is equivalent to the weight of about 6680 lbm of mass. Therefore, the support bars must be strong enough to withstand the weight of 6680 lbm hanged at one of their ends when they are held from the other end horizontally.

---

**11-29**

**Solution** An advertisement sign in the form of a rectangular block that has the same frontal area from all four sides is mounted on top of a taxicab. The increase in the annual fuel cost due to this sign is to be determined.

***Assumptions*** **1** The flow of air is steady and incompressible. **2** The car is driven 60,000 km a year at an average speed of 50 km/h. **3** The overall efficiency of the engine is 28%. **4** The effect of the sign and the taxicab on the drag coefficient of each other is negligible (no interference), and the edge effects of the sign are negligible (a crude approximation). **5** The flow is turbulent so that the tabulated value of the drag coefficient can be used.




***Properties*** The densities of air and gasoline are given to be 1.25 kg/m$^3$ and 0.75 kg/L, respectively. The heating value of gasoline is given to be 42,000 kJ/kg. The drag coefficient for a square rod for normal flow is $C_D$ = 2.2 (Table 11-1).

***Analysis*** Noting that 1 m/s = 3.6 km/h, the drag force acting on the sign is

$$F_D = C_D A \frac{\rho V^2}{2} = (2.2)(0.9 \times 0.3\,\text{m}^2)\frac{(1.25\,\text{kg/m}^3)(50/3.6\,\text{m/s})^2}{2}\left(\frac{1\,\text{N}}{1\,\text{kg}\cdot\text{m/s}^2}\right) = 71.61\,\text{N}$$

Noting that work is force times distance, the amount of work done to overcome this drag force and the required energy input for a distance of 60,000 km are

$$W_{\text{drag}} = F_D L = (71.61\,\text{N})(60{,}000\,\text{km/year}) = 4.30 \times 10^6\ \text{kJ/year}$$

$$E_{in} = \frac{W_{\text{drag}}}{\eta_{\text{car}}} = \frac{4.30 \times 10^6\ \text{kJ/year}}{0.28} = 1.54 \times 10^7\ \text{kJ/year}$$

Then the amount and cost of the fuel that supplies this much energy are

$$\text{Amont of fuel} = \frac{m_{\text{fuel}}}{\rho_{\text{fuel}}} = \frac{E_{\text{in}}/\text{HV}}{\rho_{\text{fuel}}} = \frac{(1.54 \times 10^7\ \text{kJ/year})/(42{,}000\,\text{kJ/kg})}{0.75\,\text{kg/L}} = \mathbf{489\ L/year}$$

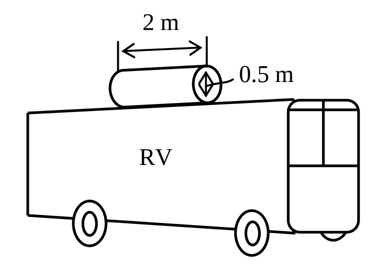
$$\text{Cost} = (\text{Amount of fuel})(\text{Unit cost}) = (489\,\text{L/year})(\$0.50/\text{L}) = \mathbf{\$245/year}$$

That is, the taxicab will use 489 L of gasoline at a cost of $245 per year to overcome the drag generated by the advertisement sign.

***Discussion*** Note that the advertisement sign increases the fuel cost of the taxicab significantly. The taxicab operator may end up losing money by installing the sign if he/she is not aware of the major increase in the fuel cost, and negotiate accordingly.

**11-30**

**Solution** The water needs of a recreational vehicle (RV) are to be met by installing a cylindrical tank on top of the vehicle. The additional power requirements of the RV at a specified speed for two orientations of the tank are to be determined.




***Assumptions*** **1** The flow of air is steady and incompressible. **2** The effect of the tank and the RV on the drag coefficient of each other is negligible (no interference). **3** The flow is turbulent so that the tabulated value of the drag coefficient can be used.

***Properties*** The drag coefficient for a cylinder corresponding to $L/D$ = 2/0.5 = 4 is $C_D$ = 0.9 when the circular surfaces of the tank face the front and back, and $C_D$ = 0.8 when the circular surfaces of the tank face the sides of the RV (Table 11-2). The density of air at the specified conditions is

$$\rho = \frac{P}{RT} = \frac{87\text{ kPa}}{(0.287\text{ kPa}\cdot\text{m}^3/\text{kg.K})(295\text{ K})} = 1.028\text{ kg/m}^3$$

***Analysis*** (*a*) The drag force acting on the tank when the circular surfaces face the front and back is

$$F_D = C_D A\frac{\rho V^2}{2} = (0.9)[\pi(0.5\text{ m})^2/4]\frac{(1.028\text{ kg/m}^3)(95\text{ km/h})^2}{2}\left(\frac{1\text{ m/s}}{3.6\text{ km/h}}\right)^2\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 63.3\text{ N}$$

Noting that power is force times velocity, the amount of additional power needed to overcome this drag force is

$$\dot{W}_{\text{drag}} = F_D V = (63.3\text{ N})(95/3.6\text{ m/s})\left(\frac{1\text{ kW}}{1000\text{ N}\cdot\text{m/s}}\right) = \mathbf{1.67\text{ kW}}$$

(*b*) The drag force acting on the tank when the circular surfaces face the sides of the RV is

$$F_D = C_D A\frac{\rho V^2}{2} = (0.8)[0.5\times 2\text{ m}^2]\frac{(1.028\text{ kg/m}^3)(95\text{ km/h})^2}{2}\left(\frac{1\text{ m/s}}{3.6\text{ km/h}}\right)^2\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 286\text{ N}$$

Then the additional power needed to overcome this drag force is

$$\dot{W}_{\text{drag}} = F_D V = (286\text{ N})(95/3.6\text{ m/s})\left(\frac{1\text{ kW}}{1000\text{ N}\cdot\text{m/s}}\right) = \mathbf{7.55\text{ kW}}$$

Therefore, the additional power needed to overcome the drag caused by the tank is 1.67 kW and 7.55 W for the two orientations indicated.

***Discussion*** Note that the additional power requirement is the lowest when the tank is installed such that its circular surfaces face the front and back of the RV. This is because the frontal area of the tank is minimum in this orientation.

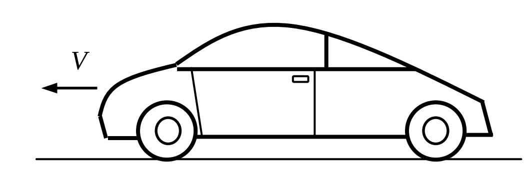
**11-31E**

**Solution** A person who normally drives at 55 mph now starts driving at 75 mph. The percentage increase in fuel consumption of the car is to be determined.

***Assumptions*** **1** The fuel consumption is approximately proportional to the drag force on a level road (as stated). **2** The drag coefficient remains the same.

***Analysis*** The drag force is proportional to the square of the velocity, and power is force times velocity. Therefore,

$$F_D = C_D A\frac{\rho V^2}{2} \quad \text{and} \quad \dot{W}_{\text{drag}} = F_D V = C_D A\frac{\rho V^3}{2}$$




Then the ratio of the drag force at $V_2$ = 75 mph to the drag force at $V_1$ = 55 mph becomes

$$\frac{F_{D2}}{F_{D1}} = \frac{V_2^2}{V_1^2} = \frac{75^2}{55^2} = \mathbf{1.86}$$ , an **increase of 86%**.

Therefore, the power to overcome the drag force and thus fuel consumption per unit time nearly doubles as a result of increasing the velocity from 55 to 75 mph.

***Discussion*** This increase appears to be large. This is because all the engine power is assumed to be used entirely to overcome drag. Still, the simple analysis above shows the strong dependence of the fuel consumption on the cruising speed of a vehicle. A better measure of fuel consumption is the amount of fuel used per unit distance (rather than per unit time). A car cruising at 55 mph will travel a distance of 55 miles in 1 hour. But a car cruising at 75 mph will travel the same distance at 55/75 = 0.733 h or 73.3% of the time. Therefore, for a given distance, the increase in fuel consumption is 1.86×0.733 = 1.36 – an increase of **36%**. This is still large, especially with the high cost of gasoline these days.

---

**11-32**

**Solution** A plastic sphere is dropped into water. The terminal velocity of the sphere in water is to be determined.

***Assumptions*** **1** The fluid flow over the sphere is laminar (to be verified). **2** The drag coefficient remains constant.

***Properties*** The density of plastic sphere is 1150 kg/m$^3$. The density and dynamic viscosity of water at 20°C are $\rho$ = 998 kg/m$^3$ and $\mu$ = 1.002×10$^{-3}$ kg/m·s, respectively. The drag coefficient for a sphere in laminar flow is $C_D$ = 0.5.

***Analysis*** The terminal velocity of a free falling object is reached when the drag force equals the weight of the solid object less the buoyancy force applied by the fluid,

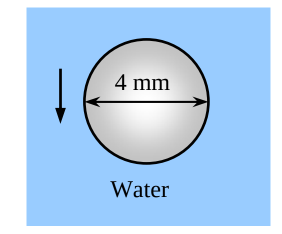
$$F_D = W - F_B \quad \text{where} \quad F_D = C_D A\frac{\rho_f V^2}{2}, \quad W = \rho_s g\mathcal{V}, \quad \text{and } F_B = \rho_f g\mathcal{V}$$

Here $A = \pi D^2/4$ is the frontal area and $\mathcal{V} = \pi D^3/6$ is the volume of the sphere. Substituting and simplifying,

$$C_D A\frac{\rho_f V^2}{2} = \rho_s g\mathcal{V} - \rho_f g\mathcal{V} \;\rightarrow\; C_D \frac{\pi D^2}{4}\frac{\rho_f V^2}{2} = (\rho_s - \rho_f) g\frac{\pi D^3}{6} \;\rightarrow\; C_D\frac{V^2}{8} = \left(\frac{\rho_s}{\rho_f} - 1\right)\frac{gD}{6}$$

Solving for $V$ and substituting,

$$V = \sqrt{\frac{4gD(\rho_s/\rho_f - 1)}{3C_D}} = \sqrt{\frac{4(9.81\text{ m/s}^2)(0.004\text{ m})(1150/998\text{-}1)}{3\times 0.5}} = \mathbf{0.126\ m/s}$$




The Reynolds number is

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(998\text{ kg/m}^3)(0.126\text{ m/s})(4\times 10^{-3}\text{ m})}{1.002\times 10^{-3}\text{ kg}\cdot\text{m/s}} = 503$$

which is less than 2×10$^5$. Therefore, the flow is laminar as assumed.

***Discussion*** This problem can be solved more accurately using a trial-and-error approach by using $C_D$ data from Fig. 11-34 (the $C_D$ value corresponding to Re = 503 is about 0.6. Repeating the calculations for this value gives 0.115 m/s for the terminal velocity.

---

**11-33**

**Solution** A semi truck is exposed to winds from its side surface. The wind velocity that will tip the truck over to its side is to be determined.

***Assumptions*** **1** The flow of air in the wind is steady and incompressible. **2** The edge effects on the semi truck are negligible (a crude approximation), and the resultant drag force acts through the center of the side surface. **3** The flow is turbulent so that the tabulated value of the drag coefficient can be used. **4** The distance between the wheels on the same axle is also 2 m. **5** The semi truck is loaded uniformly so that its weight acts through its center.

***Properties*** The density of air is given to be ρ = 1.10 kg/m$^3$. The drag coefficient for a body of rectangular cross-section corresponding to $L/D$ = 2/2 = 1 is $C_D$ = 2.2 when the wind is normal to the side surface (Table 11-2).

***Analysis*** When the truck is first tipped, the wheels on the wind-loaded side of the truck will be off the ground, and thus all the reaction forces from the ground will act on wheels on the other side. Taking the moment about an axis passing through these wheels and setting it equal to zero gives the required drag force to be

$$\sum M_{\text{wheels}} = 0 \;\rightarrow\; F_D \times (1\text{ m}) - W \times (1\text{ m}) = 0 \;\rightarrow\; F_D = W$$



Substituting, the required drag force is determined to be

$$F_D = mg = (5000\text{ kg})(9.81\text{ m/s}^2)\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 49{,}050\text{ N}$$

The wind velocity that will cause this drag force is determined to be

$$F_D = C_D A\frac{\rho V^2}{2} \;\rightarrow\; 49{,}050\text{ N} = (2.2)(2\times 8\text{ m}^2)\frac{(1.10\text{ kg/m}^3)V^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) \;\rightarrow\; V = 50.3\text{ m/s}$$

which is equivalent to a wind velocity of $V$ = 50.3×3.6 = **181 km/h.**

***Discussion*** This is very high velocity, and it can be verified easily by calculating the Reynolds number that the flow is turbulent as assumed.

## 11-34

**Solution** A bicyclist is riding his bicycle downhill on a road with a specified slope without pedaling or breaking. The terminal velocity of the bicyclist is to be determined for the upright and racing positions.

***Assumptions*** **1** The rolling resistance and bearing friction are negligible. **2** The drag coefficient remains constant. **3** The buoyancy of air is negligible.

***Properties*** The density of air is given to be $\rho$ = 1.25 kg/m$^3$. The frontal area and the drag coefficient are given to be 0.45 m$^2$ and 1.1 in the upright position, and 0.4 m$^2$ and 0.9 on the racing position.

***Analysis*** The terminal velocity of a free falling object is reached when the drag force equals the component of the total weight (bicyclist + bicycle) in the flow direction,

$$F_D = W_{\text{total}} \sin\theta \quad \rightarrow \quad C_D A \frac{\rho V^2}{2} = m_{\text{total}} g \sin\theta$$

Solving for $V$ gives $$V = \sqrt{\frac{2 g m_{\text{total}} \sin\theta}{C_D A \rho}}$$

The terminal velocities for both positions are obtained by substituting the given values:

Upright position: $$V = \sqrt{\frac{2(9.81\text{ m/s}^2)(80+15\text{ kg})\sin 12°}{1.1(0.45\text{ m}^2)(1.25\text{ kg/m}^3)}} = 25.0\text{ m/s} = \textbf{90 km/h}$$

Racing position: $$V = \sqrt{\frac{2(9.81\text{ m/s}^2)(80+15\text{ kg})\sin 12°}{0.9(0.4\text{ m}^2)(1.25\text{ kg/m}^3)}} = 29.3\text{ m/s} = \textbf{106 km/h}$$

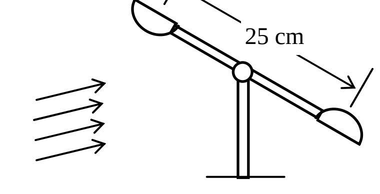


***Discussion*** Note that the position of the bicyclist has a significant effect on the drag force, and thus the terminal velocity. So it is no surprise that the bicyclists maintain the racing position during a race.

---

## 11-35

**Solution** The pivot of a wind turbine with two hollow hemispherical cups is stuck as a result of some malfunction. For a given wind speed, the maximum torque applied on the pivot is to be determined.




***Assumptions*** **1** The flow of air in the wind is steady and incompressible. **2** The air flow is turbulent so that the tabulated values of the drag coefficients can be used.

***Properties*** The density of air is given to be ρ = 1.25 kg/m$^3$. The drag coefficient for a hemispherical cup is 0.4 and 1.2 when the hemispherical and plain surfaces are exposed to wind flow, respectively.

***Analysis*** The maximum torque occurs when the cups are normal to the wind since the length of the moment arm is maximum in this case. Noting that the frontal area is $\pi D^2/4$ for both cups, the drag force acting on each cup is determined to be

*Convex side*: $$F_{D1} = C_{D1} A \frac{\rho V^2}{2} = (0.4)[\pi(0.08\text{ m})^2/4]\frac{(1.25\text{ kg/m}^3)(15\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 0.283\text{ N}$$

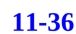
*Concave side*: $$F_{D2} = C_{D2} A \frac{\rho V^2}{2} = (1.2)[\pi(0.08\text{ m})^2/4]\frac{(1.25\text{ kg/m}^3)(15\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 0.848\text{ N}$$

The moment arm for both forces is 12.5 cm since the distance between the centers of the two cups is given to be 25 cm. Taking the moment about the pivot, the net torque applied on the pivot is determined to be

$$M_{\max} = F_{D2}L - F_{D1}L = (F_{D2} - F_{D1})L = (0.848 - 0.283\text{ N})(0.125\text{ m}) = \textbf{0.0706 Nm}$$

***Discussion*** Note that the torque varies between zero when both cups are aligned with the wind to the maximum value calculated above.

---

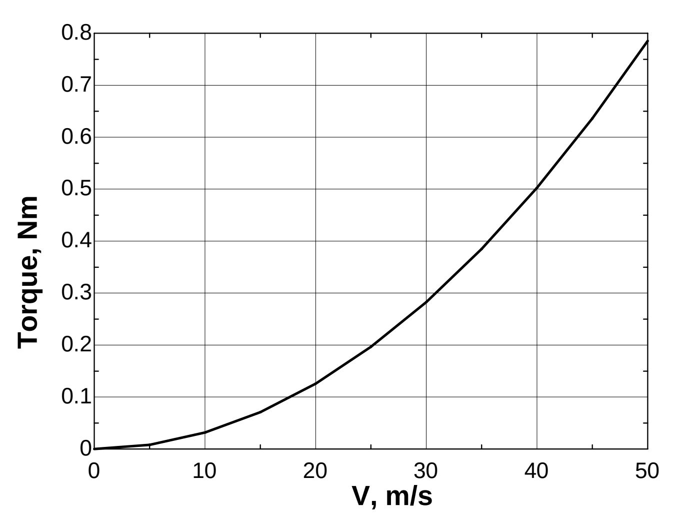


**11-36** 

**Solution** The previous problem is reconsidered. The effect of wind speed on the torque applied on the pivot as the wind speed varies from 0 to 50 m/s in increments of 5 m/s is to be investigated.

***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
CD1=0.40 "Curved bottom"
CD2=1.2 "Plain frontal area"
"rho=density(Air, T=T, P=P)" "kg/m^3"
rho=1.25 "kg/m3"
D=0.08 "m"
L=0.25 "m"
A=pi*D^2/4 "m^2"
FD1=CD1*A*(rho*V^2)/2 "N"
FD2=CD2*A*(rho*V^2)/2 "N"
FD_net=FD2-FD1
Torque=(FD2-FD1)*L/2
```




| $V$, m/s | $F_{drag,\ net}$, N | Torque, Nm |
|---|---|---|
| 0 | 0.00 | 0.000 |
| 5 | 0.06 | 0.008 |
| 10 | 0.25 | 0.031 |
| 15 | 0.57 | 0.071 |
| 20 | 1.01 | 0.126 |
| 25 | 1.57 | 0.196 |
| 30 | 2.26 | 0.283 |
| 35 | 3.08 | 0.385 |
| 40 | 4.02 | 0.503 |
| 45 | 5.09 | 0.636 |
| 50 | 6.28 | 0.785 |

***Discussion*** Since drag force grows as velocity squared, the torque also grows as velocity squared.

**11-37E**

**Solution** A spherical tank completely submerged in fresh water is being towed by a ship at a specified velocity. The required towing power is to be determined.

***Assumptions*** **1** The flow is turbulent so that the tabulated value of the drag coefficient can be used. **2** The drag of the towing bar is negligible.

***Properties*** The drag coefficient for a sphere is $C_D$ = 0.2 in turbulent flow (it is 0.5 for laminar flow). We take the density of water to be 62.4 lbm/ft$^3$.

***Analysis*** The frontal area of a sphere is $A = \pi D^2/4$. Then the drag force acting on the spherical tank is

$$F_D = C_D A\frac{\rho V^2}{2} = (0.2)[\pi(5\text{ ft})^2/4]\frac{(62.4\text{ lbm/ft}^3)(12\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = 548\text{ lbf}$$

Since power is force times velocity, the power needed to overcome this drag force during towing is

$$\dot{W}_{\text{Towing}} = \dot{W}_{\text{drag}} = F_D V = (548\text{ lbf})(12\text{ ft/s})\left(\frac{1\text{ kW}}{737.56\text{ lbf}\cdot\text{ft/s}}\right) = 8.92\text{ kW} = 12.0\text{ hp}$$

Therefore, the additional power needed to tow the tank is **12.0 hp**.

***Discussion*** Note that the towing power is proportional the cube of the velocity. Therefore, the towing power can be reduced to one-eight (which is 1.5 hp) by reducing the towing velocity by half to 6 ft/s. But the towing time will double this time for a given distance.

---

**11-38**

**Solution** The power delivered to the wheels of a car is used to overcome aerodynamic drag and rolling resistance. For a given power, the speed at which the rolling resistance is equal to the aerodynamic drag and the maximum speed of the car are to be determined.

***Assumptions*** **1** The air flow is steady and incompressible. **2** The bearing friction is negligible. **3** The drag and rolling resistance coefficients of the car are constant. **4** The car moves horizontally on a level road.

***Properties*** The density of air is given to be $\rho$ = 1.20 kg/m$^3$. The drag and rolling resistance coefficients are given to be $C_D$ = 0.32 and $C_{RR}$ = 0.04, respectively.

***Analysis*** (*a*) The rolling resistance of the car is

$$F_{RR} = C_{RR}W = 0.04(950\text{ kg})(9.81\text{ m/s}^2)\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 372.8\text{ N}$$

The velocity at which the rolling resistance equals the aerodynamic drag force is determined by setting these two forces equal to each other,

$$F_D = C_D A\frac{\rho V^2}{2} \;\rightarrow\; 372.8\text{ N} = (0.32)(1.8\text{ m}^2)\frac{(1.20\text{ kg/m}^3)V^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) \;\rightarrow\; V = 32.8\text{ m/s (or 118 km/h)}$$

(*b*) Power is force times speed, and thus the power needed to overcome drag and rolling resistance is the product of the sum of the drag force and the rolling resistance and the velocity of the car,

$$\dot{W}_{\text{total}} = \dot{W}_{\text{drag}} + \dot{W}_{\text{RR}} = (F_D + F_{RR})V = C_D A\frac{\rho V^3}{2} + F_{RR}V$$

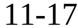
Substituting the known quantities, the maximum speed corresponding to a wheel power of 80 kW is determined to be

$$(0.32)(1.8\text{ m}^2)\frac{(1.20\text{ kg/m}^3)V^3}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) + 372.8V = 80{,}000\text{ W}\left(\frac{1\text{ N}\cdot\text{m/s}}{1\text{ W}}\right) \quad \text{or,} \quad 0.3456V^3 + 372.8V = 80{,}000$$

whose solution is $V$ = 55.56 m/s **= 200 km/h**.

***Discussion*** A net power input of 80 kW is needed to overcome rolling resistance and aerodynamic drag at a velocity of 200 km/h. About 75% of this power is used to overcome drag and the remaining 25% to overcome the rolling resistance. At much higher velocities, the fraction of drag becomes even higher as it is proportional to the cube of car velocity.

---

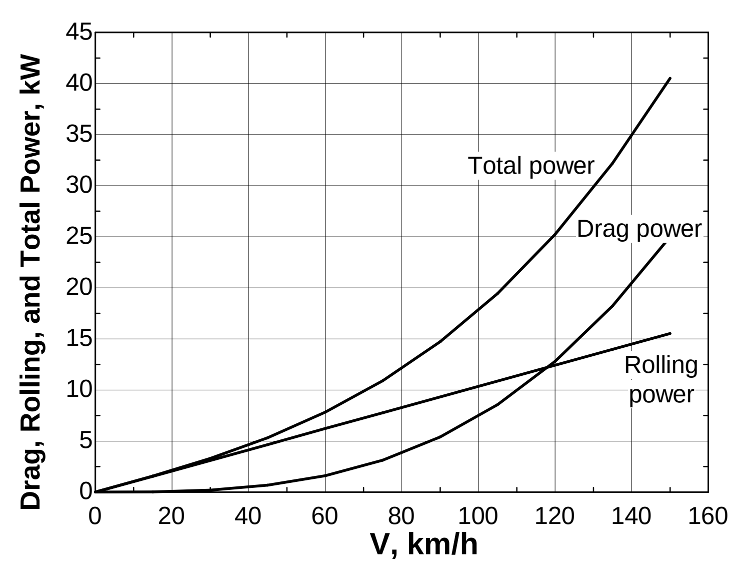


**11-39** 

**Solution** The previous problem is reconsidered. The effect of car speed on the required power to overcome (*a*) rolling resistance, (*b*) the aerodynamic drag, and (*c*) their combined effect as the car speed varies from 0 to 150 km/h in increments of 15 km/h is to be investigated.

***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
rho=1.20 "kg/m3"
C_roll=0.04
m=950 "kg"
g=9.81 "m/s2"
V=Vel/3.6 "m/s"
W=m*g
F_roll=C_roll*W
A=1.8 "m2"
C_D=0.32
F_D=C_D*A*(rho*V^2)/2 "N"
Power_RR=F_roll*V/1000 "W"
Power_Drag=F_D*V/1000 "W"
Power_Total=Power_RR+Power_Drag
```




| $V$, m/s | $W_{drag}$ kW | $W_{rolling}$, kW | $W_{total}$, $kW$ |
|---|---|---|---|
| 0 | 0.00 | 0.00 | 0.00 |
| 15 | 0.03 | 1.55 | 1.58 |
| 30 | 0.20 | 3.11 | 3.31 |
| 45 | 0.68 | 4.66 | 5.33 |
| 60 | 1.60 | 6.21 | 7.81 |
| 75 | 3.13 | 7.77 | 10.89 |
| 90 | 5.40 | 9.32 | 14.72 |
| 105 | 8.58 | 10.87 | 19.45 |
| 120 | 12.80 | 12.43 | 25.23 |
| 135 | 18.23 | 13.98 | 32.20 |
| 150 | 25.00 | 15.53 | 40.53 |

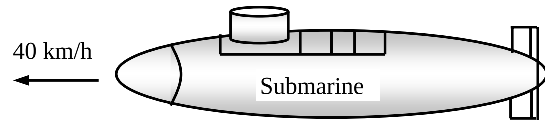
***Discussion*** Notice that the rolling power curve and drag power curve intersect at about 118 km/h (73.3 mph). So, near highway speeds, the overall power is split nearly 50% between rolling drag and aerodynamic drag.

**11-40**

**Solution** A submarine is treated as an ellipsoid at a specified length and diameter. The powers required for this submarine to cruise horizontally in seawater and to tow it in air are to be determined.

***Assumptions*** **1** The submarine can be treated as an ellipsoid. **2** The flow is turbulent. **3** The drag of the towing rope is negligible. **4** The motion of submarine is steady and horizontal.

***Properties*** The drag coefficient for an ellipsoid with $L/D = 25/5 = 5$ is $C_D = 0.1$ in turbulent flow (Table 11-2). The density of sea water is given to be 1025 kg/m$^3$. The density of air is given to be 1.30 kg/m$^3$.




***Analysis*** Noting that 1 m/s = 3.6 km/h, the velocity of the submarine is equivalent to $V = 40/3.6 = 11.11$ m/s. The frontal area of an ellipsoid is $A = \pi D^2/4$. Then the drag force acting on the submarine becomes

*In water*: $$F_D = C_D A \frac{\rho V^2}{2} = (0.1)[\pi (5\text{ m})^2/4] \frac{(1025\text{ kg/m}^3)(11.11\text{ m/s})^2}{2} \left( \frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2} \right) = 124.2\text{ kN}$$

*In air*: $$F_D = C_D A \frac{\rho V^2}{2} = (0.1)[\pi (5\text{ m})^2/4] \frac{(1.30\text{ kg/m}^3)(11.11\text{ m/s})^2}{2} \left( \frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2} \right) = 0.158\text{ kN}$$

Noting that power is force times velocity, the power needed to overcome this drag force is

*In water*: $$\dot{W}_{\text{drag}} = F_D V = (124.2\text{ kN})(11.11\text{ m/s}) \left( \frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}} \right) = \mathbf{1380\text{ kW}}$$

*In air*: $$\dot{W}_{\text{drag}} = F_D V = (0.158\text{ kN})(11.11\text{ m/s}) \left( \frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}} \right) = \mathbf{1.75\text{ kW}}$$

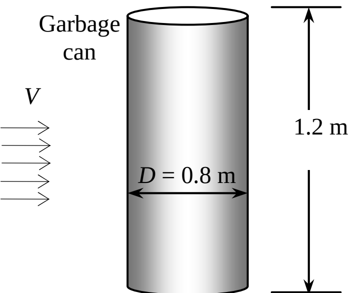
Therefore, the power required for this submarine to cruise horizontally in seawater is 1380 kW and the power required to tow this submarine in air at the same velocity is 1.75 kW.

***Discussion*** Note that the power required to move the submarine in water is about 800 times the power required to move it in air. This is due to the higher density of water compared to air (sea water is about 800 times denser than air).

**11-41**

**Solution** A garbage can is found in the morning tipped over due to high winds the night before. The wind velocity during the night when the can was tipped over is to be determined.

***Assumptions*** **1** The flow of air in the wind is steady and incompressible. **2** The ground effect on the wind and the drag coefficient is negligible (a crude approximation) so that the resultant drag force acts through the center of the side surface. **3** The garbage can is loaded uniformly so that its weight acts through its center.




***Properties*** The density of air is given to be $\rho = 1.25$ kg/m$^3$, and the average density of the garbage inside the can is given to be 150 kg/m$^3$. The drag coefficient of the garbage can is given to be 0.7.

***Analysis*** The volume of the garbage can and the weight of the garbage are

$$V = [\pi D^2/4]H = [\pi(0.80\text{ m})^2/4](1.2\text{ m}) = 0.6032\text{ m}^2$$

$$W = mg = \rho g V = (150\text{ kg/m}^3)(9.81\text{ m/s}^2)(0.6032\text{ m}^3)\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 887.6\text{ N}$$

When the garbage can is first tipped, the edge on the wind-loaded side of the can will be off the ground, and thus all the reaction forces from the ground will act on the other side. Taking the moment about an axis passing through the contact point and setting it equal to zero gives the required drag force to be

$$\sum M_{\text{contact}} = 0 \ \rightarrow \ F_D \times (H/2) - W \times (D/2) = 0 \ \rightarrow \ F_D = \frac{WD}{H} = \frac{(887.6\text{ N})(0.80\text{ m})}{1.2\text{ m}} = 591.7\text{ N}$$

Noting that the frontal area is *DH*, the wind velocity that will cause this drag force is determined to be

$$F_D = C_D A\frac{\rho V^2}{2} \ \rightarrow \ 591.7\text{ N} = (0.7)[1.2\times 0.80\text{ m}^2]\frac{(1.25\text{ kg/m}^3)V^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) \ \rightarrow \ V = 37.5\text{ m/s}$$

which is equivalent to a wind velocity of $V = 37.5\times 3.6$ **= 135 km/h.**

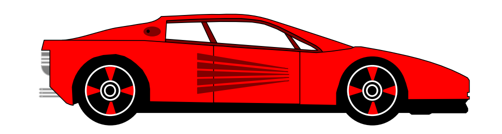
***Discussion*** The analysis above shows that under the stated assumptions, the wind velocity at some moment exceeded 135 km/h. But we cannot tell how high the wind velocity has been. Such analysis and predictions are commonly used in forensic engineering.

**11-42E**

**Solution** The drag coefficient of a sports car increases when the sunroof is open, and it requires more power to overcome aerodynamic drag. The additional power consumption of the car when the sunroof is opened is to be determined at two different velocities.

***Assumptions*** **1** The car moves steadily at a constant velocity on a straight path. **2** The effect of velocity on the drag coefficient is negligible.

***Properties*** The density of air is given to be 0.075 lbm/ft$^3$. The drag coefficient of the car is given to be $C_D$ = 0.32 when the sunroof is closed, and $C_D$ = 0.41 when it is open.



***Analysis*** (*a*) Noting that 1 mph = 1.4667 ft/s and that power is force times velocity, the drag force acting on the car and the power needed to overcome it at 35 mph are:

*Open sunroof*: $$F_{D1} = 0.32(18\text{ ft}^2)\frac{(0.075\text{ lbm/ft}^3)(35\times 1.4667\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = 17.7\text{ lbf}$$

$$\dot{W}_{\text{drag1}} = F_{D1}V = (17.7\text{ lbf})(35\times 1.4667\text{ ft/s})\left(\frac{1\text{ kW}}{737.56\text{ lbf}\cdot\text{ft/s}}\right) = 1.23\text{ kW}$$

*Closed sunroof*: $$F_{D2} = 0.41(18\text{ ft}^2)\frac{(0.075\text{ lbm/ft}^3)(35\times 1.4667\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = 22.7\text{ lbf}$$

$$\dot{W}_{\text{drag2}} = F_{D2}V = (22.7\text{ lbf})(35\times 1.4667\text{ ft/s})\left(\frac{1\text{ kW}}{737.56\text{ lbf}\cdot\text{ft/s}}\right) = 1.58\text{ kW}$$

Therefore, the additional power required for this car when the sunroof is open is

$$\dot{W}_{\text{extra}} = \dot{W}_{\text{drag2}} - \dot{W}_{\text{drag2}} = 1.58 - 1.23 = \mathbf{0.350\ kW} \quad \text{(at 35 mph)}$$

(*b*) We now repeat the calculations for 70 mph:

*Open sunroof*: $$F_{D1} = 0.32(18\text{ ft}^2)\frac{(0.075\text{ lbm/ft}^3)(70\times 1.4667\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = 70.7\text{ lbf}$$

$$\dot{W}_{\text{drag1}} = F_{D1}V = (70.7\text{ lbf})(70\times 1.4667\text{ ft/s})\left(\frac{1\text{ kW}}{737.56\text{ lbf}\cdot\text{ft/s}}\right) = 9.82\text{ kW}$$

*Closed sunroof*: $$F_{D2} = 0.41(18\text{ ft}^2)\frac{(0.075\text{ lbm/ft}^3)(70\times 1.4667\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = 90.6\text{ lbf}$$

$$\dot{W}_{\text{drag2}} = F_{D2}V = (90.6\text{ lbf})(70\times 1.4667\text{ ft/s})\left(\frac{1\text{ kW}}{737.56\text{ lbf}\cdot\text{ft/s}}\right) = 12.6\text{ kW}$$

Therefore, the additional power required for this car when the sunroof is open is

$$\dot{W}_{\text{extra}} = \dot{W}_{\text{drag2}} - \dot{W}_{\text{drag2}} = 12.6 - 9.82 = \mathbf{2.78\ kW} \quad \text{(at 70 mph)}$$

***Discussion*** Note that the additional drag caused by open sunroof is 0.35 kW at 35 mph, and 2.78 kW at 70 mph, which is an increase of 8 folds when the velocity is doubled. This is expected since the power consumption to overcome drag is proportional to the cube of velocity.

## Flow over Flat Plates

**11-43C**

**Solution** We are to discuss the fluid property responsible for the development of a boundary layer.

***Analysis*** The **fluid viscosity** is responsible for the development of the velocity boundary layer. Velocity forces the boundary layer closer to the wall. Therefore, the higher the velocity (and thus Reynolds number), the lower the thickness of the boundary layer.

***Discussion*** All fluids have viscosity – a measure of frictional forces in a fluid. There is no such thing as an *inviscid fluid*, although there are *regions*, called *inviscid flow regions*, in which viscous effects are negligible.

**11-44C**

**Solution** We are to define and discuss the friction coefficient for flow over a flat plate.

***Analysis*** The *friction coefficient* represents **the resistance to fluid flow over a flat plate**. It is proportional to the drag force acting on the plate. The drag coefficient for a flat surface is equivalent to the mean friction coefficient.

***Discussion*** In flow over a flat plate aligned with the flow, there is no pressure (form) drag – only friction drag.

**11-45C**

**Solution** We are to discuss how the local skin friction coefficient changes with position along a flat plate in laminar flow.

***Analysis*** The local friction coefficient **decreases with downstream distance** in laminar flow over a flat plate.

***Discussion*** At the front of the plate, the boundary layer is very thin, and thus the shear stress at the wall is large. As the boundary layer grows downstream, however, the boundary layer grows in size, decreasing the wall shear stress.

**11-46C**

**Solution** We are to define and discuss the average skin friction coefficient over a flat plate.

***Analysis*** The *average friction coefficient* in flow over a flat plate is determined by **integrating the local friction coefficient over the entire length of the plate**, and then dividing it by the length of the plate. Or, it can be determined experimentally by measuring the drag force, and dividing it by the dynamic pressure.

***Discussion*** For the case of a flat plate aligned with the flow, there is no pressure drag, only skin friction drag. Thus, the average friction coefficient is the same as the drag coefficient. This is not true for other body shapes.

**11-47E**

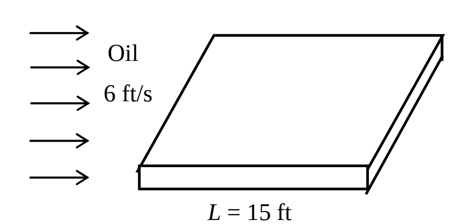
**Solution** Light oil flows over a flat plate. The total drag force per unit width of the plate is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The critical Reynolds number is $Re_{cr} = 5 \times 10^5$. **3** The surface of the plate is smooth.

***Properties*** The density and kinematic viscosity of light oil at 75°F are $\rho = 55.3$ lbm/ft$^3$ and $\nu = 7.751\times10^{-3}$ ft$^2$/s.

***Analysis*** Noting that $L$ = 15 ft, the Reynolds number at the end of the plate is

$$\mathrm{Re}_L = \frac{VL}{\nu} = \frac{(6\ \mathrm{ft/s})(15\ \mathrm{ft})}{7.751\times10^{-3}\ \mathrm{ft^2/s}} = 1.161\times10^4$$




which is less than the critical Reynolds number. Thus we have *laminar flow* over the entire plate, and the average friction coefficient is determined from

$$C_f = 1.328\,\mathrm{Re}_L^{-0.5} = 1.328\times(1.161\times10^4)^{-0.5} = 0.01232$$

Noting that the pressure drag is zero and thus $C_D = C_f$ for a flat plate, the drag force acting on the top surface of the plate per unit width becomes

$$F_D = C_f A\frac{\rho V^2}{2} = 0.01232\times(15\times1\ \mathrm{ft^2})\frac{(56.8\ \mathrm{lbm/ft^3})(6\ \mathrm{ft/s})^2}{2}\left(\frac{1\ \mathrm{lbf}}{32.2\ \mathrm{lbm\cdot ft/s^2}}\right) = \mathbf{5.87\ lbf}$$

The total drag force acting on the entire plate can be determined by multiplying the value obtained above by the width of the plate.

***Discussion*** The force per unit width corresponds to the weight of a mass of 5.87 lbm. Therefore, a person who applies an equal and opposite force to the plate to keep it from moving will feel like he or she is using as much force as is necessary to hold a 5.87 lbm mass from dropping.

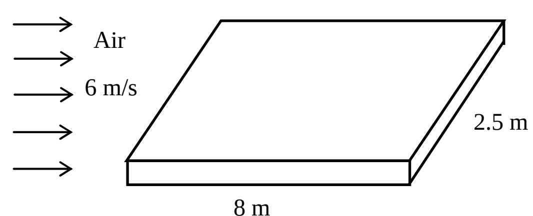
**11-48**

**Solution** Air flows over a plane surface at high elevation. The drag force acting on the top surface of the plate is to be determined for flow along the two sides of the plate.

***Assumptions*** **1** The flow is steady and incompressible. **2** The critical Reynolds number is $Re_{cr} = 5\times10^5$. **3** Air is an ideal gas. **4** The surface of the plate is smooth.

***Properties*** The dynamic viscosity is independent of pressure, and for air at 25°C it is $\mu = 1.849\times10^{-5}$ kg/m·s. The air density at 25°C = 298 K and 83.4 kPa is

$$\rho = \frac{P}{RT} = \frac{83.4\text{ kPa}}{(0.287\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(298\text{ K})} = 0.9751\text{ kg/m}^3$$




***Analysis*** (a) If the air flows parallel to the 8 m side, the Reynolds number becomes

$$Re_L = \frac{\rho VL}{\mu} = \frac{(0.9751\text{ kg/m}^3)(6\text{ m/s})(8\text{ m})}{1.849\times10^{-5}\text{ kg/m}\cdot\text{s}} = 2.531\times10^6$$

which is greater than the critical Reynolds number. Thus we have combined laminar and turbulent flow, and the friction coefficient is determined to be

$$C_f = \frac{0.074}{Re_L^{1/5}} - \frac{1742}{Re_L} = \frac{0.074}{(2.531\times10^6)^{1/5}} - \frac{1742}{2.531\times10^6} = 0.003189$$

Noting that the pressure drag is zero and thus $C_D = C_f$ for a flat plate, the drag force acting on the top surface of the plate becomes

$$F_D = C_f A\frac{\rho V^2}{2} = 0.003189\times(8\times2.5\text{ m}^2)\frac{(0.9751\text{ kg/m}^3)(6\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = \mathbf{1.12\ N}$$

(b) If the air flows parallel to the 2.5 m side, the Reynolds number is

$$Re_L = \frac{\rho VL}{\mu} = \frac{(0.9751\text{ kg/m}^3)(6\text{ m/s})(2.5\text{ m})}{1.849\times10^{-5}\text{ kg/m}\cdot\text{s}} = 7.910\times10^5$$

which is greater than the critical Reynolds number. Thus we have combined laminar and turbulent flow, and the friction coefficient is determined to be

$$C_f = \frac{0.074}{Re_L^{1/5}} - \frac{1742}{Re_L} = \frac{0.074}{(7.910\times10^5)^{1/5}} - \frac{1742}{7.910\times10^5} = 0.002691$$

Then the drag force acting on the top surface of the plate becomes

$$F_D = C_f A\frac{\rho V^2}{2} = 0.002691\times(8\times2.5\text{ m}^2)\frac{(0.9751\text{ kg/m}^3)(6\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = \mathbf{0.94\ N}$$

***Discussion*** Note that the drag force is proportional to density, which is proportional to the pressure. Therefore, the altitude has a major influence on the drag force acting on a surface. Commercial airplanes take advantage of this phenomenon and cruise at high altitudes where the air density is much lower to save fuel.

**11-49**

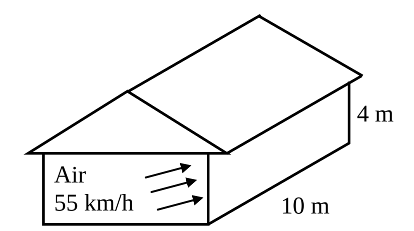
**Solution** Wind is blowing parallel to the side wall of a house. The drag force acting on the wall is to be determined for two different wind velocities.

***Assumptions*** **1** The flow is steady and incompressible. **2** The critical Reynolds number is $Re_{cr} = 5\times10^5$. **3** Air is an ideal gas. **4** The wall surface is smooth (the actual wall surface is usually very rough). **5** The wind blows parallel to the wall.

***Properties*** The density and kinematic viscosity of air at 1 atm and 5°C are $\rho = 1.269$ kg/m$^3$ and $\nu = 1.382\times10^{-5}$ m$^2$/s .

***Analysis*** The Reynolds number is

$$Re_L = \frac{VL}{\nu} = \frac{\left[(55/3.6)\ \text{m/s}\right](10\ \text{m})}{1.382\times10^{-5}\ \text{m}^2/\text{s}} = 1.105\times10^7$$




which is greater than the critical Reynolds number. Thus we have combined laminar and turbulent flow, and the friction coefficient is

$$C_f = \frac{0.074}{Re_L^{1/5}} - \frac{1742}{Re_L} = \frac{0.074}{(1.105\times10^7)^{1/5}} - \frac{1742}{1.105\times10^7} = 0.002730$$

Noting that the pressure drag is zero and thus $C_D = C_f$ for a flat plate, the drag force acting on the wall surface is

$$F_D = C_f A\frac{\rho V^2}{2} = 0.00273\times(10\times4\ \text{m}^2)\frac{(1.269\ \text{kg/m}^3)(55/3.6\ \text{m/s})^2}{2}\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = \mathbf{16.2\ N}$$

(b) When the wind velocity is doubled to 110 km/h, the Reynolds number becomes

$$Re_L = \frac{VL}{\nu} = \frac{\left[(110/3.6)\ \text{m/s}\right](10\ \text{m})}{1.382\times10^{-5}\ \text{m}^2/\text{s}} = 2.211\times10^7$$

which is greater than the critical Reynolds number. Thus we have combined laminar and turbulent flow, and the friction coefficient and the drag force become

$$C_f = \frac{0.074}{Re_L^{1/5}} - \frac{1742}{Re_L} = \frac{0.074}{(2.211\times10^7)^{1/5}} - \frac{1742}{2.211\times10^7} = 0.002435$$

$$F_D = C_f A\frac{\rho V^2}{2} = 0.002435\times(10\times4\ \text{m}^2)\frac{(1.269\ \text{kg/m}^3)(110/3.6\ \text{m/s})^2}{2}\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = \mathbf{57.7\ N}$$

Treating flow over the side wall of a house as flow over a flat plate is not quite realistic. When flow hits a bluff body like a house, it separates at the sharp corner and a separation bubble exists over most of the side panels of the house. Therefore, flat plat boundary layer equations are not appropriate for this problem, and the entire house should considered in the solution instead.

***Discussion*** Note that the actual drag will probably be much higher since the wall surfaces are typically very rough. Also, we can solve this problem using the turbulent flow relation (instead of the combined laminar-turbulent flow relation) without much loss in accuracy. Finally, the drag force nearly quadruples when the velocity is doubled. This is expected since the drag force is proportional to the square of the velocity, and the effect of velocity on the friction coefficient is small.



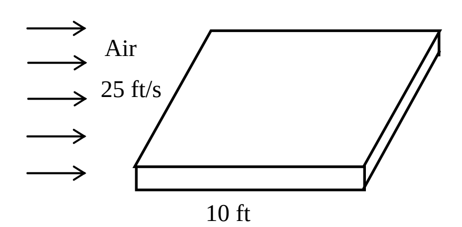
**11-50E** 

**Solution** Air flows over a flat plate. The local friction coefficients at intervals of 1 ft is to be determined and plotted against the distance from the leading edge.

***Assumptions*** **1** The flow is steady and incompressible. **2** The critical Reynolds number is $Re_{cr} = 5\times10^5$. **3** Air is an ideal gas. **4** The surface of the plate is smooth.

***Properties*** The density and kinematic viscosity of air at 1 atm and 70°F are $\rho = 0.07489$ lbm/ft$^3$ and $\nu = 0.5913$ ft$^2$/h = $1.643\times10^{-4}$ ft$^2$/s .

***Analysis*** For the first 1 ft interval, the Reynolds number is




$$\text{Re}_L = \frac{VL}{\nu} = \frac{(25\text{ ft/s})(1\text{ ft})}{1.643\times10^{-4}\text{ ft}^2\text{/s}} = 1.522\times10^5$$

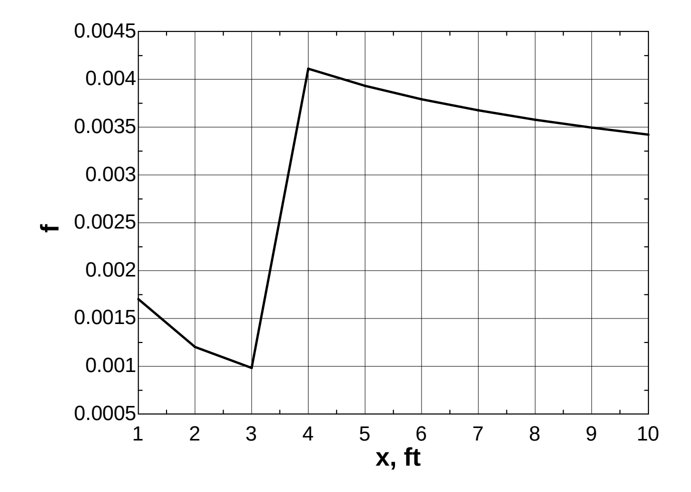
which is less than the critical value of $5\times10^5$. Therefore, the flow is laminar. The local friction coefficient is

$$C_{f,x} = \frac{0.664}{\text{Re}^{0.5}} = \frac{0.664}{(1.522\times10^5)^{0.5}} = 0.001702$$

We repeat calculations for all 1-ft intervals. The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
rho=0.07489 "lbm/ft3"
nu=0.5913/3600 "ft2/s"
V=25
"Local Re and C_f"
Re=x*V/nu
"f=0.664/Re^0.5"
f=0.059/Re^0.2
```

| *x*, ft | Re | $C_f$ |
|---|---|---|
| 1 | 1.522E+05 | 0.001702 |
| 2 | 3.044E+05 | 0.001203 |
| 3 | 4.566E+05 | 0.000983 |
| 4 | 6.088E+05 | 0.004111 |
| 5 | 7.610E+05 | 0.003932 |
| 6 | 9.132E+05 | 0.003791 |
| 7 | 1.065E+06 | 0.003676 |
| 8 | 1.218E+06 | 0.003579 |
| 9 | 1.370E+06 | 0.003496 |
| 10 | 1.522E+06 | 0.003423 |




***Discussion*** Note that the Reynolds number exceeds the critical value for $x > 3$ ft, and thus the flow is turbulent over most of the plate. For $x > 3$ ft, we used $C_f = 0.074/\text{Re}_L^{1/5}$-$1742/\text{Re}_L$ for friction coefficient. Note that $C_f$ decreases with Re in both laminar and turbulent flows.

**11-51**

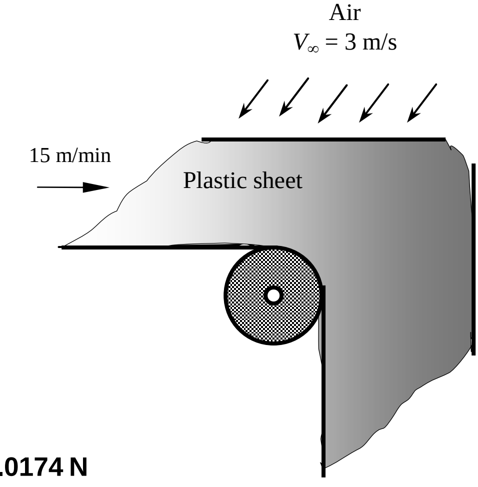
**Solution** Air flows on both sides of a continuous sheet of plastic. The drag force air exerts on the plastic sheet in the direction of flow is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The critical Reynolds number is $\text{Re}_{cr} = 5\times10^5$. **3** Air is an ideal gas. **4** Both surfaces of the plastic sheet are smooth. **5** The plastic sheet does not vibrate and thus it does not induce turbulence in air flow.

***Properties*** The density and kinematic viscosity of air at 1 atm and 60°C are $\rho = 1.059$ kg/m$^3$ and $\nu = 1.896\times10^{-5}$ m$^2$/s .

***Analysis*** The length of the cooling section is

$$L = V_{\text{sheet}}\Delta t = [(15/60)\text{ m/s}](2\text{ s}) = 0.5\text{ m}$$




The Reynolds number is

$$\text{Re}_L = \frac{VL}{\nu} = \frac{(3\text{ m/s})(1.2\text{ m})}{1.896\times10^{-5}\text{ m}^2\text{/s}} = 1.899\times10^5$$

which is less than the critical Reynolds number. Thus the flow is laminar. The area on both sides of the sheet exposed to air flow is

$$A = 2wL = 2(1.2\text{ m})(0.5\text{ m}) = 1.2\text{ m}^2$$

Then the friction coefficient and the drag force become

$$C_f = \frac{1.328}{\text{Re}_L^{0.5}} = \frac{1.328}{(1.899\times10^5)^{0.5}} = 0.003048$$

$$F_D = C_f A\frac{\rho V^2}{2} = (0.003048)(1.2\text{ m}^2)\frac{(1.059\text{ kg/m}^3)(3\text{ m/s})^2}{2} = \mathbf{0.0174\text{ N}}$$

***Discussion*** Note that the Reynolds number remains under the critical value, and thus the flow remains laminar over the entire plate. In reality, the flow may be turbulent because of the motion of the plastic sheet.

---

**11-52**

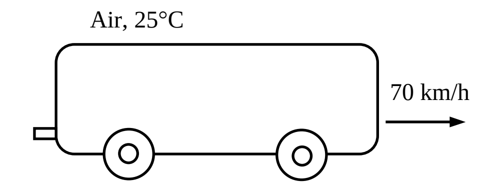
**Solution** A train is cruising at a specified velocity. The drag force acting on the top surface of a passenger car of the train is to be determined.

***Assumptions*** **1** The air flow is steady and incompressible. **2** The critical Reynolds number is $\text{Re}_{cr} = 5\times10^5$. **3** Air is an ideal gas. **4** The top surface of the train is smooth (in reality it can be rough). **5** The air is calm (no significant winds).

***Properties*** The density and kinematic viscosity of air at 1 atm and 25°C are $\rho = 1.184$ kg/m$^3$ and $\nu = 1.562\times10^{-5}$ m$^2$/s .

***Analysis*** The Reynolds number is

$$\text{Re}_L = \frac{VL}{\nu} = \frac{\left[(70/3.6)\text{ m/s}\right](8\text{ m})}{1.562\times10^{-5}\text{ m}^2\text{/s}} = 9.959\times10^6$$




which is greater than the critical Reynolds number. Thus we have combined laminar and turbulent flow, and the friction coefficient is determined to be

$$C_f = \frac{0.074}{\text{Re}_L^{1/5}} - \frac{1742}{\text{Re}_L} = \frac{0.074}{(9.959\times10^6)^{1/5}} - \frac{1742}{9.959\times10^6} = 0.002774$$

Noting that the pressure drag is zero and thus $C_D = C_f$ for a flat plate, the drag force acting on the surface becomes

$$F_D = C_f A\frac{\rho V^2}{2} = 0.002774\times(8\times3.2\text{ m}^2)\frac{(1.184\text{ kg/m}^3)(70/3.6\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = \mathbf{15.9\text{ N}}$$

***Discussion*** Note that we can solve this problem using the turbulent flow relation (instead of the combined laminar-turbulent flow relation) without much loss in accuracy since the Reynolds number is much greater than the critical value. Also, the actual drag force will probably be greater because of the surface roughness effects.

---

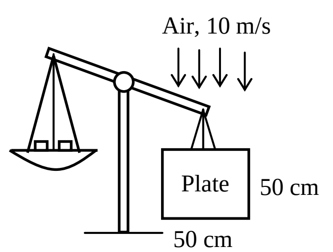
**11-53**

**Solution** The weight of a thin flat plate exposed to air flow on both sides is balanced by a counterweight. The mass of the counterweight that needs to be added in order to balance the plate is to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The critical Reynolds number is $\mathrm{Re}_{cr} = 5\times10^5$. **3** Air is an ideal gas. **4** The surfaces of the plate are smooth.

***Properties*** The density and kinematic viscosity of air at 1 atm and 25°C are $\rho = 1.184$ kg/m$^3$ and $\nu = 1.562\times10^{-5}$ m$^2$/s .

***Analysis*** The Reynolds number is




$$\mathrm{Re}_L = \frac{VL}{\nu} = \frac{(10\text{ m/s})(0.5\text{ m})}{1.562\times10^{-5}\text{ m}^2/\text{s}} = 3.201\times10^5$$

which is less than the critical Reynolds number of $5\times10^5$ . Therefore the flow is laminar. The average friction coefficient, drag force and the corresponding mass are

$$C_f = \frac{1.328}{\mathrm{Re}_L^{0.5}} = \frac{1.328}{(3.201\times10^5)^{0.5}} = 0.002347$$

$$F_D = C_f A\frac{\rho V^2}{2} = (0.002347)[(2\times0.5\times0.5)\text{ m}^2]\frac{(1.184\text{ kg/m}^3)(10\text{ m/s})^2}{2} = 0.0695\text{ kg}\cdot\text{m/s}^2 = 0.0695\text{ N}$$

The mass whose weight is 0.0695 N is

$$m = \frac{F_D}{g} = \frac{0.0695\text{ kg.m/s}^2}{9.81\text{ m/s}^2} = \mathbf{0.0071\,kg} = \mathbf{7.1\,g}$$

Therefore, the mass of the counterweight must be 7.1 g to counteract the drag force acting on the plate.

***Discussion*** Note that the apparatus described in this problem provides a convenient mechanism to measure drag force and thus drag coefficient.

---

**11-54**

**Solution** Laminar flow of a fluid over a flat plate is considered. The change in the drag force is to be determined when the free-stream velocity of the fluid is doubled.

***Analysis*** For the laminar flow of a fluid over a flat plate the drag force is given by

$$F_{D1} = C_f A\frac{\rho V^2}{2} \quad \text{where} \quad C_f = \frac{1.328}{\mathrm{Re}^{0.5}}$$

Therefore

$$F_{D1} = \frac{1.328}{\mathrm{Re}^{0.5}} A\frac{\rho V^2}{2}$$

Substituting Reynolds number relation, we get

$$F_{D1} = \frac{1.328}{\left(\frac{VL}{\upsilon}\right)^{0.5}} A\frac{\rho V^2}{2} = 0.664V^{3/2}A\frac{\upsilon^{0.5}}{L^{0.5}}$$

When the free-stream velocity of the fluid is doubled, the new value of the drag force on the plate becomes

$$F_{D2} = \frac{1.328}{\left(\frac{(2V)L}{\upsilon}\right)^{0.5}} A\frac{\rho(2V)^2}{2} = 0.664(2V)^{3/2}A\frac{\upsilon^{0.5}}{L^{0.5}}$$

The ratio of drag forces corresponding to $V$ and $2V$ is

$$\frac{F_{D2}}{F_{D1}} = \frac{(2V)^{3/2}}{V^{3/2}} = \mathbf{2^{3/2}} = \mathbf{2.83}$$

***Discussion*** Note that the drag force increases almost three times in laminar flow when the fluid velocity is doubled.

---

**11-55E**

**Solution** A refrigeration truck is traveling at a specified velocity. The drag force acting on the top and side surfaces of the truck and the power needed to overcome it are to be determined.

***Assumptions*** **1** The process is steady and incompressible. **2** The airflow over the entire outer surface is turbulent because of constant agitation. **3** Air is an ideal gas. **4** The top and side surfaces of the truck are smooth (in reality they can be rough). **5** The air is calm (no significant winds).

***Properties*** The density and kinematic viscosity of air at 1 atm and 80°F are $\rho$ = 0.07350 lbm/ft$^3$ and $\nu$ = 0.6110 ft$^2$/s = 1.697×10$^{-4}$ ft$^2$/s .

***Analysis*** The Reynolds number is

$$\text{Re}_L = \frac{VL}{\nu} = \frac{\left[(65\times 1.4667)\text{ ft/s}\right](20\text{ ft})}{1.697\times 10^{-4}\text{ ft}^2/\text{s}} = 1.124\times 10^7$$

20 ft

8 ft

Refrigeration truck

Air
65 mph

The air flow over the entire outer surface is assumed to be turbulent. The friction coefficient becomes

$$C_f = \frac{0.074}{\text{Re}_L^{1/5}} = \frac{0.074}{(1.124\times 10^7)^{1/5}} = 0.002878$$

The area of the top and side surfaces of the truck is

$$A = A_{top} + 2A_{side} = 9\times 20 + 2\times 8\times 20 = 500\text{ ft}^2$$

Noting that the pressure drag is zero and thus $C_D = C_f$ for a plane surface, the drag force acting on these surfaces becomes

$$F_D = C_f A\frac{\rho V^2}{2} = 0.002878\times(500\text{ ft}^2)\frac{(0.07350\text{ lbm/ft}^3)(65\times 1.4667\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = \mathbf{14.9\text{ lbf}}$$

Noting that power is force times velocity, the power needed to overcome this drag force is

$$\dot{W}_{drag} = F_D V = (14.9\text{ lbf})(65\times 1.4667\text{ ft/s})\left(\frac{1\text{ kW}}{737.56\text{ lbf}\cdot\text{ft/s}}\right) = \mathbf{1.93\text{ kW}}$$

***Discussion*** Note that the calculated drag force (and the power required to overcome it) is very small. This is not surprising since the drag force for blunt bodies is almost entirely due to pressure drag, and the friction drag is practically negligible compared to the pressure drag.

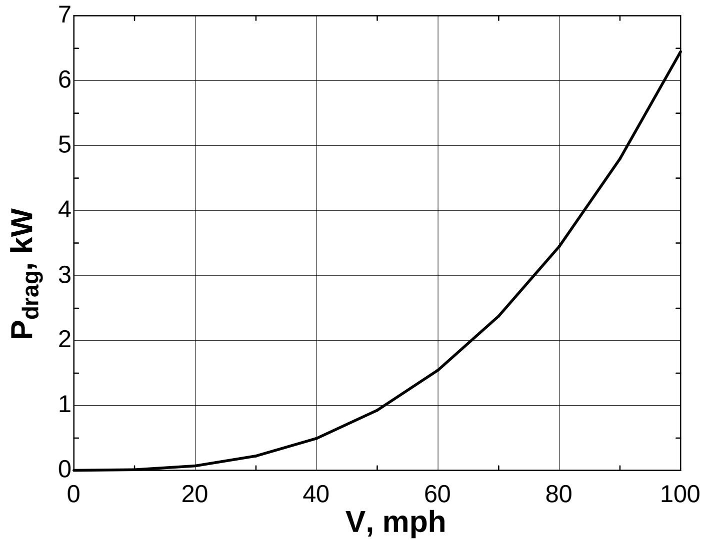

**11-56E**



**Solution** The previous problem is reconsidered. The effect of truck speed on the total drag force acting on the top and side surfaces, and the power required to overcome as the truck speed varies from 0 to 100 mph in increments of 10 mph is to be investigated.

***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
rho=0.07350 "lbm/ft3"
nu=0.6110/3600 "ft2/s"
V=Vel*1.4667 "ft/s"
L=20 "ft"
W=2*8+9
A=L*W
Re=L*V/nu
Cf=0.074/Re^0.2
g=32.2 "ft/s2"
F=Cf*A*(rho*V^2)/2/32.2 "lbf"
Pdrag=F*V/737.56 "kW"
```




| *V*, mph | Re | $F_{drag}$, lbf | $P_{drag}$, *kW* |
|---|---|---|---|
| 0 | 0 | 0.00 | 0.000 |
| 10 | 1.728E+06 | 0.51 | 0.010 |
| 20 | 3.457E+06 | 1.79 | 0.071 |
| 30 | 5.185E+06 | 3.71 | 0.221 |
| 40 | 6.913E+06 | 6.23 | 0.496 |
| 50 | 8.642E+06 | 9.31 | 0.926 |
| 60 | 1.037E+07 | 12.93 | 1.542 |
| 70 | 1.209E+07 | 17.06 | 2.375 |
| 80 | 1.382E+07 | 21.69 | 3.451 |
| 90 | 1.555E+07 | 26.82 | 4.799 |
| 100 | 1.728E+07 | 32.42 | 6.446 |

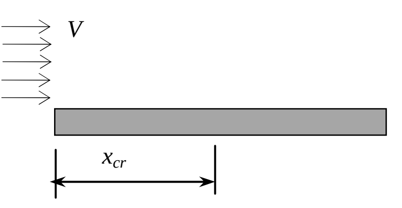
***Discussion*** The required power increases rapidly with velocity – in fact, as velocity *cubed*.

**11-57**

**Solution** Air is flowing over a long flat plate with a specified velocity. The distance from the leading edge of the plate where the flow becomes turbulent, and the thickness of the boundary layer at that location are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The critical Reynolds number is $Re_{cr} = 5\times10^5$. **3** Air is an ideal gas. **4** The surface of the plate is smooth.

***Properties*** The density and kinematic viscosity of air at 1 atm and 25°C are $\rho = 1.184$ kg/m$^3$ and $\nu = 1.562\times10^{-5}$ m$^2$/s .




***Analysis*** The critical Reynolds number is given to be $Re_{cr} = 5\times10^5$. The distance from the leading edge of the plate where the flow becomes turbulent is the distance $x_{cr}$ where the Reynolds number becomes equal to the critical Reynolds number,

$$\mathrm{Re}_{cr} = \frac{Vx_{cr}}{\nu} \quad \rightarrow \quad x_{cr} = \frac{\nu \mathrm{Re}_{cr}}{V} = \frac{(1.562\times10^{-5}\ \mathrm{m^2/s})(5\times10^5)}{8\ \mathrm{m/s}} = \mathbf{0.976\ m}$$

The thickness of the boundary layer at that location is obtained by substituting this value of $x$ into the laminar boundary layer thickness relation,

$$\delta_{v,x} = \frac{4.91x}{\mathrm{Re}_x^{1/2}} \quad \rightarrow \quad \delta_{v,cr} = \frac{4.91x_{cr}}{\mathrm{Re}_{cr}^{1/2}} = \frac{4.91(0.976\ \mathrm{m})}{(5\times10^5)^{1/2}} = 0.00678\ \mathrm{m} = \mathbf{0.678\ cm}$$

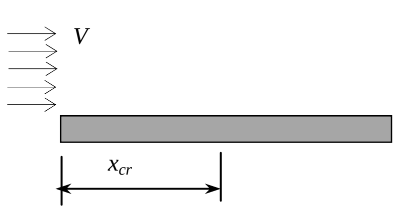
***Discussion*** When the flow becomes turbulent, the boundary layer thickness starts to increase, and the value of its thickness can be determined from the boundary layer thickness relation for turbulent flow.

**11-58**

**Solution** Water is flowing over a long flat plate with a specified velocity. The distance from the leading edge of the plate where the flow becomes turbulent, and the thickness of the boundary layer at that location are to be determined.

***Assumptions*** **1** The flow is steady and incompressible. **2** The critical Reynolds number is $Re_{cr} = 5\times10^5$. **3** The surface of the plate is smooth.

***Properties*** The density and dynamic viscosity of water at 1 atm and 25°C are $\rho = 997$ kg/m$^3$ and $\mu = 0.891\times10^{-3}$ kg/m·s.




***Analysis*** The critical Reynolds number is given to be $Re_{cr} = 5\times10^5$. The distance from the leading edge of the plate where the flow becomes turbulent is the distance $x_{cr}$ where the Reynolds number becomes equal to the critical Reynolds number,

$$\mathrm{Re}_{cr} = \frac{\rho V x_{cr}}{\mu} \quad \rightarrow \quad x_{cr} = \frac{\mu \mathrm{Re}_{cr}}{\rho V} = \frac{(0.891\times10^{-3}\ \mathrm{kg/m\cdot s})(5\times10^5)}{(997\ \mathrm{kg/m^3})(8\ \mathrm{m/s})} = \mathbf{0.056\ m}$$

The thickness of the boundary layer at that location is obtained by substituting this value of $x$ into the laminar boundary layer thickness relation,

$$\delta_{v,x} = \frac{5x}{\mathrm{Re}_x^{1/2}} \quad \rightarrow \quad \delta_{v,cr} = \frac{4.91x_{cr}}{\mathrm{Re}_{cr}^{1/2}} = \frac{4.91(0.056\ \mathrm{m})}{(5\times10^5)^{1/2}} = 0.00039\ \mathrm{m} = \mathbf{0.39\ mm}$$

Therefore, the flow becomes turbulent after about 5.6 cm from the leading edge of the plate, and the thickness of the boundary layer at that location is 0.39 mm.

***Discussion*** When the flow becomes turbulent, the boundary layer thickness starts to increase, and the value of its thickness can be determined from the boundary layer thickness relation for turbulent flow.

## Flow across Cylinders and Spheres

**11-59C**

**Solution** We are to discuss why the drag coefficient suddenly drops when the flow becomes turbulent.

***Analysis*** Turbulence **moves the fluid separation point further back on the rear of the body, reducing the size of the wake, and thus the magnitude of the pressure drag** (which is the dominant mode of drag). As a result, the drag coefficient suddenly drops. In general, turbulence increases the drag coefficient for flat surfaces, but the drag coefficient usually remains constant at high Reynolds numbers when the flow is turbulent.

***Discussion*** The sudden drop in drag is sometimes referred to as the *drag crisis*.

**11-60C**

**Solution** We are to discuss how pressure drag and friction drag differ in flow over blunt bodies.

***Analysis*** *Friction drag* is due to the **shear stress at the surface** whereas *pressure drag* is due to **the pressure differential between the front and back sides of the body** because of the wake that is formed in the rear.

***Discussion*** For a blunt or bluff body, pressure drag is usually greater than friction drag, while for a well-streamlined body, the opposite is true. For the case of a flat plate aligned with the flow, *all* of the drag is friction drag.

**11-61C**

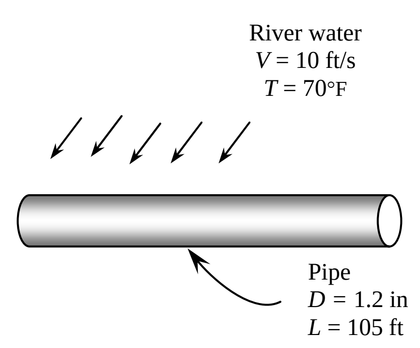
**Solution** We are to discuss why flow separation is delayed in turbulent flow over circular cylinders.

***Analysis*** Flow separation in flow over a cylinder is delayed in turbulent flow **because of the extra mixing due to random fluctuations and the transverse motion**.

***Discussion*** As a result of the turbulent mixing, a turbulent boundary layer can resist flow separation better than a laminar boundary layer can, under otherwise similar conditions.

**11-62E**

**Solution** A pipe is crossing a river while remaining completely immersed in water. The drag force exerted on the pipe by the river is to be determined.




***Assumptions*** **1** The outer surface of the pipe is smooth so that Fig. 11-34 can be used to determine the drag coefficient. **2** Water flow in the river is steady. **3** The turbulence in water flow in the river is not considered. **4** The direction of water flow is normal to the pipe.

***Properties*** The density and dynamic viscosity of water at 70°F are $\rho$ = 62.30 lbm/ft$^3$ and $\mu$ = 2.36 lbm/ft·h = 6.556×10$^{-4}$ lbm/ft·s.

***Analysis*** Noting that $D$ = 1.2 in = 0.1 ft, the Reynolds number for flow over the pipe is

$$\text{Re} = \frac{VD}{\nu} = \frac{\rho VD}{\mu} = \frac{(62.30\ \text{lbm/ft}^3)(10\ \text{ft/s})(0.1\ \text{ft})}{6.556\times10^{-4}\ \text{lbm/ft}\cdot\text{s}} = 9.50\times10^4$$

The drag coefficient corresponding to this value is, from Fig. 11-34, $C_D$ = 1.1. Also, the frontal area for flow past a cylinder is $A = LD$. Then the drag force acting on the cylinder becomes

$$F_D = C_D A\frac{\rho V^2}{2} = 1.1\times(105\times0.1\ \text{ft}^2)\frac{(62.30\ \text{lbm/ft}^3)(10\ \text{ft/s})^2}{2}\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right) = \mathbf{1120\ lbf}$$

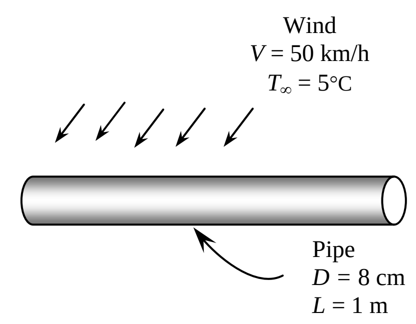
***Discussion*** Note that this force is equivalent to the weight of a 1120 lbm mass. Therefore, the drag force the river exerts on the pipe is equivalent to hanging a mass of 1120 lbm on the pipe supported at its ends 70 ft apart. The necessary precautions should be taken if the pipe cannot support this force. Also, the fluctuations in water flow may reduce the drag coefficients by inducing turbulence and delaying flow separation.

---

**11-63**

**Solution** A pipe is exposed to high winds. The drag force exerted on the pipe by the winds is to be determined.

***Assumptions*** **1** The outer surface of the pipe is smooth so that Fig. 11-34 can be used to determine the drag coefficient. **2** Air flow in the wind is steady and incompressible. **3** The turbulence in the wind is not considered. **4**The direction of wind is normal to the pipe.




***Properties*** The density and kinematic viscosity of air at 1 atm and 5°C are $\rho$ = 1.269 kg/m$^3$ and $\nu$ = 1.382×10$^{-5}$ m$^2$/s.

***Analysis*** Noting that $D$ = 0.08 m and 1 m/s = 3.6 km/h, the Reynolds number for flow over the pipe is

$$\text{Re} = \frac{VD}{\nu} = \frac{(50/3.6\ \text{m/s})(0.08\ \text{m})}{1.382\times10^{-5}\ \text{m}^2/\text{s}} = 0.8040\times10^5$$

The drag coefficient corresponding to this value is, from Fig. 11-34, $C_D$ = 1.0. Also, the frontal area for flow past a cylinder is $A = LD$. Then the drag force becomes

$$F_D = C_D A\frac{\rho V^2}{2} = 1.0(1\times0.08\ \text{m}^2)\frac{(1.269\ \text{kg/m}^3)(50/3.6\ \text{m/s})^2}{2}\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = \mathbf{9.79\ N}\ \ \text{(per m length)}$$

***Discussion*** Note that the drag force acting on a unit length of the pipe is equivalent to the weight of 1 kg mass. The total drag force acting on the entire pipe can be obtained by multiplying the value obtained by the pipe length. It should be kept in mind that wind turbulence may reduce the drag coefficients by inducing turbulence and delaying flow separation.

---

**11-64E**

**Solution** A person extends his uncovered arms into the windy air outside. The drag force exerted on both arms by the wind is to be determined.

***Assumptions*** **1** The surfaces of the arms are smooth so that Fig. 11-34 can be used to determine the drag coefficient. **2** Air flow in the wind is steady and incompressible. **3** The turbulence in the wind is not considered. **4**The direction of wind is normal to the arms. **5** The arms can be treated as 2-ft-long and 3-in.-diameter cylinders with negligible end effects.

***Properties*** The density and kinematic viscosity of air at 1 atm and 60°F are $\rho = 0.07633$ lbm/ft$^3$ and $\nu = 0.5718$ ft$^2$/h = $1.588\times10^{-4}$ ft$^2$/s.

Air
60°F, 20 mph
Arm

***Analysis*** Noting that $D$ = 3 in = 0.25 ft and 1 mph = 1.4667 ft/s, the Reynolds number for flow over the arm is

$$\text{Re} = \frac{VD}{\nu} = \frac{(20\times1.4667\text{ ft/s})(0.25\text{ ft})}{1.588\times10^{-4}\text{ ft}^2\text{/s}} = 4.618\times10^4$$

The drag coefficient corresponding to this value is, from Fig. 11-34, $C_D$ = 1.0. Also, the frontal area for flow past a cylinder is $A = LD$. Then the total drag force acting on *both* arms becomes

$$F_D = C_D A\frac{\rho V^2}{2} = 1.0\times(2\times2\times0.25\text{ ft}^2)\frac{(0.07633\text{ lbm/ft}^3)(20\times1.4667\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = \mathbf{1.02\ lbf}$$

***Discussion*** Note that this force is equivalent to the weight of 1 lbm mass. Therefore, the drag force the wind exerts on the arms of this person is equivalent to hanging 0.5 lbm of mass on each arm. Also, it should be kept in mind that the wind turbulence and the surface roughness may affect the calculated result significantly.

---

**11-65**

**Solution** Wind is blowing across the wire of a transmission line. The drag force exerted on the wire by the wind is to be determined.

***Assumptions*** **1** The wire surfaces are smooth so that Fig. 11-34 can be used to determine the drag coefficient. **2** Air flow in the wind is steady and incompressible. **3** The turbulence in the wind is not considered. **4**The direction of wind is normal to the wire.

***Properties*** The density and kinematic viscosity of air at 1 atm and 15°C are $\rho = 1.225$ kg/m$^3$ and $\nu = 1.470\times10^{-5}$ m$^2$/s.

Wind
$V$ = 40 km/h
$T$ = 15°C
Transmission wire,
$D$ = 0.6 cm
$L$ = 120 m

***Analysis*** Noting that $D$ = 0.006 m and 1 m/s = 3.6 km/h, the Reynolds number for the flow is

$$\text{Re} = \frac{VD}{\nu} = \frac{(40/3.6\text{ m/s})(0.006\text{ m})}{1.470\times10^{-5}\text{ m}^2\text{/s}} = 4.535\times10^3$$

The drag coefficient corresponding to this value is, from Fig. 11-34, $C_D$ = 1.0. Also, the frontal area for flow past a cylinder is $A = LD$. Then the drag force becomes

$$F_D = C_D A\frac{\rho V^2}{2} = 1.0(120\times0.006\text{ m}^2)\frac{(1.225\text{ kg/m}^3)(40/3.6\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = \mathbf{54.4\ N}$$

Therefore, the drag force acting on the wire is 54.4 N, which is equivalent to the weight of about 5.4 kg mass hanging on the wire.

***Discussion*** It should be kept in mind that wind turbulence may reduce the drag coefficients by inducing turbulence and delaying flow separation.

---

**11-66**

**Solution** Spherical hail is falling freely in the atmosphere. The terminal velocity of the hail in air is to be determined.

***Assumptions*** **1** The surface of the hail is smooth so that Fig. 11-34 can be used to determine the drag coefficient. **2** The variation of the air properties with altitude is negligible. **3** The buoyancy force applied by air to hail is negligible since $\rho_{air} << \rho_{hail}$ (besides, the uncertainty in the density of hail is greater than the density of air). **4** Air flow over the hail is steady and incompressible when terminal velocity is established. **5** The atmosphere is calm (no winds or drafts).

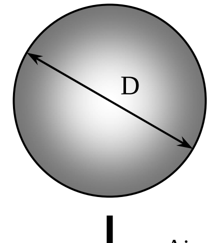
***Properties*** The density and kinematic viscosity of air at 1 atm and 5°C are $\rho$ = 1.269 kg/m$^3$ and $\nu$ = 1.382×10$^{-5}$ m$^2$/s. The density of hail is given to be 910 kg/m$^3$.

***Analysis*** The terminal velocity of a free falling object is reached when the drag force equals the weight of the solid object less the buoyancy force applied by the fluid, which is negligible in this case,

$$F_D = W - F_B \quad \text{where} \quad F_D = C_D A \frac{\rho_f V^2}{2}, \quad W = mg = \rho_s g V = \rho_s g(\pi D^3/6), \quad \text{and} \quad F_B \cong 0$$

and $A = \pi D^2/4$ is the frontal area. Substituting and simplifying,

$$C_D A \frac{\rho_f V^2}{2} = W \;\rightarrow\; C_D \frac{\pi D^2}{4} \frac{\rho_f V^2}{2} = \rho_s g \frac{\pi D^3}{6} \;\rightarrow\; C_D \rho_f V^2 = \rho_s g \frac{4D}{3}$$




Solving for $V$ and substituting,

$$V = \sqrt{\frac{4g\rho_s D}{3C_D \rho_f}} = \sqrt{\frac{4(9.81\text{ m/s}^2)(910\text{ kg/m}^3)(0.008\text{ m})}{3C_D(1.269\text{ kg/m}^3)}} \;\rightarrow\; V = \frac{8.662}{\sqrt{C_D}} \qquad (1)$$

The drag coefficient $C_D$ is to be determined from Fig. 11-34, but it requires the Reynolds number which cannot be calculated since we do not know velocity. Therefore, the solution requires a trial-error approach. First we express the Reynolds number as

$$\text{Re} = \frac{VD}{\nu} = \frac{V(0.008\text{ m})}{1.382\times 10^{-5}\text{ m}^2/\text{s}} \;\rightarrow\; \text{Re} = 578.9V \qquad (2)$$

Now we choose a velocity in m/s, calculate the Re from Eq. 2, read the corresponding $C_D$ from Fig. 11-34, and calculate $V$ from Eq. 1. Repeat calculations until the assumed velocity matches the calculated velocity. With this approach the terminal velocity is determined to be

$V$ = **13.7 m/s**

The corresponding Re and $C_D$ values are Re = 7930 and $C_D$ = 0.40. Therefore, the velocity of hail will remain constant when it reaches the terminal velocity of 13.7 m/s = 49 km/h.

***Discussion*** The simple analysis above gives us a reasonable value for the terminal velocity. A more accurate answer can be obtained by a more detailed (and complex) analysis by considering the variation of air properties with altitude, and by considering the uncertainty in the drag coefficient (a hail is not necessarily spherical and smooth).

**11-67**

**Solution** A spherical dust particle is suspended in the air at a fixed point as a result of an updraft air motion. The magnitude of the updraft velocity is to be determined using Stokes law.

***Assumptions*** **1** The Reynolds number is low (at the order of 1) so that Stokes law is applicable (to be verified). **2** The updraft is steady and incompressible. **3** The buoyancy force applied by air to the dust particle is negligible since $\rho_{air} << \rho_{dust}$ (besides, the uncertainty in the density of dust is greater than the density of air). (We will solve the problem without utilizing this assumption for generality).

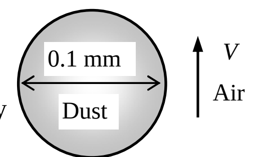
***Properties*** The density of dust is given to be $\rho_s$ = 2.1 g/cm$^3$ = 2100 kg/m$^3$. The density and dynamic viscosity of air at 1 atm and 25°C are $\rho_f$ = 1.184 kg/m$^3$ and $\mu$ = 1.849×10$^{-5}$ kg/m·s.

***Analysis*** The terminal velocity of a free falling object is reached (or the suspension of an object in a flow stream is established) when the drag force equals the weight of the solid object less the buoyancy force applied by the surrounding fluid,

$$F_D = W - F_B \quad \text{where} \quad F_D = 3\pi\mu VD \text{ (Stokes law)}, \quad W = \rho_s g\mathcal{V}, \quad \text{and } F_B = \rho_f g\mathcal{V}$$

Here $\mathcal{V} = \pi D^3/6$ is the volume of the sphere. Substituting,

$$3\pi\mu VD = \rho_s g\mathcal{V} - \rho_f g\mathcal{V} \quad \rightarrow \quad 3\pi\mu VD = (\rho_s - \rho_f)g\frac{\pi D^3}{6}$$




Solving for the velocity $V$ and substituting the numerical values, the updraft velocity is determined to be

$$V = \frac{gD^2(\rho_s - \rho_f)}{18\mu} = \frac{(9.81\text{ m/s}^2)(0.0001\text{ m})^2(2100 - 1.184)\text{ kg/m}^3}{18(1.849\times10^{-5}\text{ kg/m}\cdot\text{s})} = \mathbf{0.619\text{ m/s}}$$

The Reynolds number in this case is

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(1.184\text{ kg/m}^3)(0.619\text{ m/s})(0.0001\text{ m})}{1.849\times10^{-5}\text{ kg}\cdot\text{m/s}} = 4.0$$

which is in the order of 1. Therefore, the creeping flow idealization and thus Stokes law is applicable, and the value calculated is valid.

***Discussion*** Flow separation starts at about Re = 10. Therefore, Stokes law can be used as an approximation for Reynolds numbers up to this value, but this should be done with care.

**11-68**

**Solution** Dust particles that are unsettled during high winds rise to a specified height, and start falling back when things calm down. The time it takes for the dust particles to fall back to the ground and their velocity are to be determined using Stokes law.

***Assumptions*** **1** The Reynolds number is low (at the order of 1) so that Stokes law is applicable (to be verified). **2** The atmosphere is calm during fall back (no winds or drafts). **3** The initial transient period during which the dust particle accelerates to its terminal velocity is negligible. **4** The buoyancy force applied by air to the dust particle is negligible since $\rho_{air} << \rho_{dust}$ (besides, the uncertainty in the density of dust is greater than the density of air). (We will solve this problem without utilizing this assumption for generality).

***Properties*** The density of dust is given to be $\rho_s$ = 1.8 g/cm$^3$ = 1800 kg/m$^3$. The density and dynamic viscosity of air at 1 atm and 15°C are $\rho_f$ = 1.225 kg/m$^3$ and $\mu$ = 1.802×10$^{-5}$ kg/m·s.

***Analysis*** The terminal velocity of a free falling object is reached when the drag force equals the weight of the solid object less the buoyancy force applied by the surrounding fluid,

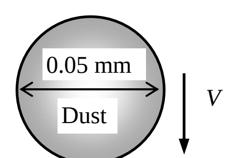
$$F_D = W - F_B \quad \text{where} \quad F_D = 3\pi\mu VD \text{ (Stokes law)}, \quad W = \rho_s g\mathcal{V}, \quad \text{and} \quad F_B = \rho_f g\mathcal{V}$$

Here $\mathcal{V} = \pi D^3/6$ is the volume of the sphere. Substituting,

$$3\pi\mu VD = \rho_s g\mathcal{V} - \rho_f g\mathcal{V} \quad \rightarrow \quad 3\pi\mu VD = (\rho_s - \rho_f)g\frac{\pi D^3}{6}$$

Solving for the velocity $V$ and substituting the numerical values, the terminal velocity is determined to be

$$V = \frac{gD^2(\rho_s - \rho_f)}{18\mu} = \frac{(9.81\text{ m/s}^2)(5\times10^{-5}\text{ m})^2(1800\text{-}1.225)\text{ kg/m}^3}{18(1.802\times10^{-5}\text{ kg/m}\cdot\text{s})} = \mathbf{0.136\text{ m/s}}$$




Then the time it takes for the dust particle to travel 350 m at this velocity becomes

$$\Delta t = \frac{L}{V} = \frac{350\text{ m}}{0.136\text{ m/s}} = 2573\text{ s} = \mathbf{42.9\text{ min}}$$

The Reynolds number is

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(1.225\text{ kg/m}^3)(0.136\text{ m/s})(5\times10^{-5}\text{ m})}{1.802\times10^{-5}\text{ kg}\cdot\text{m/s}} = 0.46$$

which is in the order of 1. Therefore, the creeping flow idealization and thus Stokes law is applicable.

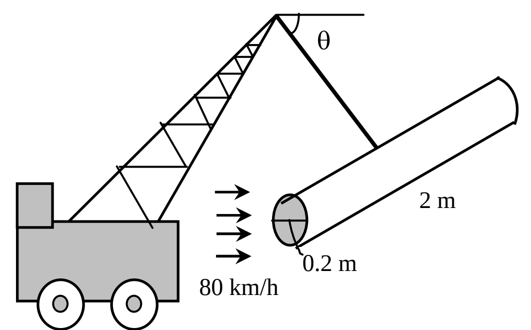
***Discussion*** Note that the dust particle reaches a terminal velocity of 0.136 m/s, and it takes about an hour to fall back to the ground. The presence of drafts in air may significantly increase the settling time.

**11-69** **[***Also solved using EES on enclosed DVD***]**

**Solution** A cylindrical log suspended by a crane is subjected to normal winds. The angular displacement of the log and the tension on the cable are to be determined.

***Assumptions*** **1** The surfaces of the log are smooth so that Fig. 11-34 can be used to determine the drag coefficient (not a realistic assumption). **2** Air flow in the wind is steady and incompressible. **3** The turbulence in the wind is not considered. **4**The direction of wind is normal to the log, which always remains horizontal. **5** The end effects of the log are negligible. **6** The weight of the cable and the drag acting on it are negligible. **7** Air is an ideal gas.

***Properties*** The dynamic viscosity of air at 5°C (independent of pressure) is $\mu$ = 1.754×10$^{-5}$ kg/m·s. Then the density and kinematic viscosity of air are calculated to be




$$\rho = \frac{P}{RT} = \frac{88\text{ kPa}}{(0.287\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(278\text{ K})} = 1.103\text{ kg/m}^3$$

$$\nu = \frac{\mu}{\rho} = \frac{1.754\times10^{-5}\text{ kg/m}\cdot\text{s}}{1.103\text{ kg/m}^3} = 1.590\times10^{-5}\text{ m}^2/\text{s}$$

***Analysis*** Noting that $D$ = 0.2 m and 1 m/s = 3.6 km/h, the Reynolds number is

$$\text{Re} = \frac{VD}{\nu} = \frac{(40/3.6\text{ m/s})(0.2\text{ m})}{1.590\times10^{-5}\text{ m}^2/\text{s}} = 1.398\times10^5$$

The drag coefficient corresponding to this value is, from Fig. 11-34, $C_D$ = 1.2. Also, the frontal area for flow past a cylinder is $A = LD$. Then the total drag force acting on the log becomes

$$F_D = C_D A\frac{\rho V^2}{2} = 1.2(2\times0.2\text{ m}^2)\frac{(1.103\text{ kg/m}^3)(40/3.6\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 32.7\text{ N}$$

The weight of the log is

$$W = mg = \rho g V = \rho g\frac{\pi D^2 L}{4} = (513\text{ kg/m}^3)(9.81\text{ m/s}^2)\frac{\pi(0.2\text{ m})^2(2\text{ m})}{4}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 316\text{ N}$$

Then the resultant force acting on the log and the angle it makes with the horizontal become

$$F_{\log} = R = \sqrt{W^2 + F_D^2} = \sqrt{32.7^2 + 316^2} = \mathbf{318\ N}$$

$$\tan\theta = \frac{W}{F_D} = \frac{316}{32.7} = 9.66 \quad\rightarrow\quad \theta = \mathbf{84°}$$

Drawing a free body diagram of the log and doing a force balance will show that the magnitude of the tension on the cable must be equal to the resultant force acting on the log. Therefore, the tension on the cable is 318 N and the cable makes 84° with the horizontal.

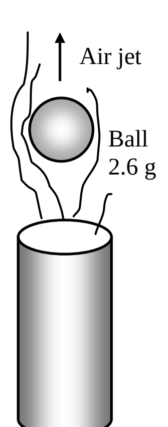
***Discussion*** Note that the wind in this case has rotated the cable by 6° from its vertical position, and increased the tension action on it somewhat. At very high wind speeds, the increase in the cable tension can be very significant, and wind loading must always be considered in bodies exposed to high winds.

**11-70**

**Solution** A ping-pong ball is suspended in air by an upward air jet. The velocity of the air jet is to be determined, and the phenomenon that the ball returns to the center of the air jet after a disturbance is to be explained.

***Assumptions*** **1** The surface of the ping-pong ball is smooth so that Fig. 11-34 can be used to determine the drag coefficient. **2** Air flow over the ball is steady and incompressible.

***Properties*** The density and kinematic viscosity of air at 1 atm and 25°C are $\rho$ = 1.184 kg/m$^3$ and $\nu$ = 1.562×10$^{-5}$ m$^2$/s.




***Analysis*** The terminal velocity of a free falling object is reached when the drag force equals the weight of the solid object less the buoyancy force applied by the fluid,

$$F_D = W - F_B \quad \text{where} \quad F_D = C_D A \frac{\rho_f V^2}{2}, \quad W = mg, \quad \text{and } F_B = \rho_f g V$$

Here $A = \pi D^2/4$ is the frontal area and $V = \pi D^3/6$ is the volume of the sphere. Also,

$$W = mg = (0.0026 \text{ kg})(9.81 \text{ m/s}^2) = 0.0255 \text{ kg}\cdot\text{m/s}^2 = 0.0255 \text{ N}$$

$$F_B = \rho_f g \frac{\pi D^3}{6} = (1.184 \text{ kg/m}^3)(9.81 \text{ m/s}^2)\frac{\pi(0.038 \text{ m})^3}{6} = 0.000334 \text{ kg}\cdot\text{m/s}^2 = 0.000334 \text{ N}$$

Substituting and solving for $V$**,**

$$C_D \frac{\pi D^2}{4}\frac{\rho_f V^2}{2} = W - F_B \;\rightarrow\; V = \sqrt{\frac{8(W-F_B)}{\pi D^2 C_D \rho_f}} = \sqrt{\frac{8(0.0255-0.000334) \text{ kg}\cdot\text{m/s}^2)}{\pi(0.038 \text{ m})^2 C_D (1.184 \text{ kg/m}^3)}} \;\rightarrow\; V = \frac{6.122}{\sqrt{C_D}} \quad (1)$$

The drag coefficient $C_D$ is to be determined from Fig. 11-34, but it requires the Reynolds number which cannot be calculated since we do not know velocity. Therefore, the solution requires a trial-error approach. First we express the Reynolds number as

$$\text{Re} = \frac{VD}{\nu} = \frac{V(0.038 \text{ m})}{1.562\times10^{-5} \text{ m}^2/\text{s}} \quad \rightarrow \quad \text{Re} = 2433V \qquad (2)$$

Now we choose a velocity in m/s, calculate the Re from Eq. 2, read the corresponding $C_D$ from Fig. 11-34, and calculate $V$ from Eq. 1. Repeat calculations until the assumed velocity matches the calculated velocity. With this approach the velocity of the fluid jet is determined to be

$V$ = **9.3 m/s**

The corresponding Re and $C_D$ values are Re = 22,600 and $C_D$ = 0.43. Therefore, the ping-pong ball will remain suspended in the air jet when the air velocity reaches 9.3 m/s = 33.5 km/h.

***Discussion***

**1** If the ball is pushed to the side by a finger, the ball will come back to the center of the jet (instead of falling off) due to the Bernoulli effect. In the core of the jet the velocity is higher, and thus the pressure is lower relative to a location away from the jet.

**2** Note that this simple apparatus can be used to determine the drag coefficients of certain object by simply measuring the air velocity, which is easy to do.

**3** This problem can also be solved roughly by taking $C_D$ = 0.5 from Table 11-2 for a sphere in laminar flow, and then verifying that the flow is laminar.

## Lift

### 11-71C

**Solution** We are to discuss why the contribution of viscous effects to lift of airfoils is usually negligible.

***Analysis*** The contribution of viscous effects to lift is usually negligible for airfoils since **the wall shear is nearly parallel to the surfaces of such devices and thus nearly normal to the direction of lift**.

***Discussion*** However, viscous effects *are* extremely important for airfoils at high angles of attack, since the viscous effects near the wall (in the boundary layer) cause the flow to separate and the airfoil to stall, losing significant lift.

### 11-72C

**Solution** We are to discuss the lift and drag on a symmetrical airfoil at zero angle of attack.

***Analysis*** When air flows past a *symmetrical* airfoil at zero angle of attack, (*a*) **the lift is zero**, but (*b*) **the drag acting on the airfoil is nonzero**.

***Discussion*** In this case, because of symmetry, there is no lift, but there is still skin friction drag, along with a small amount of pressure drag.

### 11-73C

**Solution** We are to discuss the lift and drag on a nonsymmetrical airfoil at zero angle of attack.

***Analysis*** When air flows past a *nonsymmetrical* airfoil at zero angle of attack, **both the (*a*) lift and (*b*) drag acting on the airfoil are nonzero**.

***Discussion*** Because of the lack of symmetry, the flow is different on the top and bottom surfaces of the airfoil, leading to lift. There is drag too, just as there is drag even on a symmetrical airfoil.

### 11-74C

**Solution** We are to discuss the lift and drag on a symmetrical airfoil at $5^o$ angle of attack.

***Analysis*** When air flows past a symmetrical airfoil at an angle of attack of 5°, **both the (*a*) lift and (*b*) drag acting on the airfoil are nonzero**.

***Discussion*** Because of the lack of symmetry with respect to the free-stream flow, the flow is different on the top and bottom surfaces of the airfoil, leading to lift. There is drag too, just as there is drag even at zero angle of attack.

### 11-75C

**Solution** We are to define and discuss stall.

***Analysis*** The **decrease of lift with an increase in the angle of attack** is called *stall*. When the flow separates over nearly the entire upper half of the airfoil, the lift is reduced dramatically (the separation point is near the leading edge). Stall is caused by **flow separation and the formation of a wide wake region over the top surface of the airfoil**. Commercial aircraft are not allowed to fly at velocities near the stall velocity **for safety reasons**. Airfoils stall at high angles of attack (flow cannot negotiate the curve around the leading edge). **If a plane stalls, it loses mush of its lift, and it can crash**.

***Discussion*** At angles of attack above the stall angle, the drag also increases significantly.

**11-76C**

**Solution** We are to discuss which increases at a greater rate – lift or drag – with increasing angle of attack.

***Analysis*** Both the lift and the drag of an airfoil increase with an increase in the angle of attack, but in general, **the lift increases at a much higher rate than does the drag**.

***Discussion*** In other words, the lift-to-drag ratio increases with increasing angle of attack – at least up to the stall angle.

---

**11-77C**

**Solution** We are to why flaps are used on aircraft during takeoff and landing.

***Analysis*** Flaps are used at the leading and trailing edges of the wings of large aircraft during takeoff and landing to alter the shape of the wings to **maximize lift and to enable the aircraft to land or takeoff at low speeds**. **An aircraft can take off or land without flaps, but it can do so at very high velocities**, which is undesirable during takeoff and landing.

***Discussion*** In simple terms, the planform area of the wing increases as the flaps are deployed. Thus, even if the lift coefficient were to remain constant, the actual lift would still increase. In fact, however, flaps *increase the lift coefficient* as well, leading to even further increases in lift. Lower takeoff and landing speeds lead to shorter runway length requirements.

---

**11-78C**

**Solution** We are to discuss how flaps affect the lift and drag of airplane wings.

***Analysis*** Flaps **increase both the lift and the drag of the wings**. But the increase in drag during takeoff and landing is not much of a concern because of the relatively short time periods involved. This is the penalty we pay willingly to take off and land at safe speeds.

***Discussion*** Note, however, that the engine must operate at nearly full power during takeoff to overcome the large drag.

---

**11-79C**

**Solution** We are to discuss the effect of wing tip vortices on drag and lift.

***Analysis*** The effect of wing tip vortices is to **increase drag (induced drag) and to decrease lift**. This effect is also due to the *downwash*, which **causes an effectively smaller angle of attack**.

***Discussion*** Induced drag is a three-dimensional effect; there is no induced drag on a 2-D airfoil since there are no tips.

---

**11-80C**

**Solution** We are to discuss induced drag and how to minimize it.

***Analysis*** *Induced drag* is **the additional drag caused by the tip vortices**. The tip vortices have a lot of kinetic energy, all of which is wasted and is ultimately dissipated as heat in the air downstream. **Induced drag can be reduced by using long and narrow wings, and by modifying the geometry of the wing tips**.

***Discussion*** Birds are designed with feathers that fan out at the tips of their wings in order to reduce induced drag.

---

**11-81C**

**Solution** We are to discuss the lift on a spinning and non-spinning ball.

***Analysis*** When air is flowing past a spherical ball, **the lift exerted on the ball is zero if the ball is not spinning**, and **it is nonzero if the ball is spinning about an axis normal to the free stream velocity** (no lift is generated if the ball is spinning about an axis parallel to the free stream velocity).

***Discussion*** In the parallel spinning case, however, a *side force* would be generated (e.g., a curve ball).

---

## 11-82

**Solution** A tennis ball is hit with a backspin. It is to be determined if the ball will fall or rise after being hit.

***Assumptions*** **1** The outer surface of the ball is smooth enough for Fig. 11-53 to be applicable. **2** The ball is hit horizontally so that it starts its motion horizontally.

4200 rpm

92 km/h

***Properties*** The density and kinematic viscosity of air at 1 atm and 25°C are $\rho$ = 1.184 kg/m$^3$ and $\nu$ = 1.562×10$^{-5}$ m$^2$/s.

***Analysis*** The ball is hit horizontally, and thus it would normally fall under the effect of gravity without the spin. The backspin will generate a lift, and the ball will rise if the lift is greater than the weight of the ball. The lift can be determined from

$$F_L = C_L A \frac{\rho V^2}{2}$$

where $A$ is the frontal area of the ball, $A = \pi D^2/4$. The regular and angular velocities of the ball are

$$V = (92 \text{ km/h})\left(\frac{1 \text{ m/s}}{3.6 \text{ km/h}}\right) = 25.56 \text{ m/s} \qquad \text{and} \qquad \omega = (4200 \text{ rev/min})\left(\frac{2\pi \text{ rad}}{1 \text{ rev}}\right)\left(\frac{1 \text{ min}}{60 \text{ s}}\right) = 440 \text{ rad/s}$$

Then,

$$\frac{\omega D}{2V} = \frac{(440 \text{ rad/s})(0.064 \text{ m})}{2(25.56 \text{ m/s})} = 0.551 \text{ rad}$$

From Fig. 11-53, the lift coefficient corresponding to this value is $C_L$ = 0.11. Then the lift acting on the ball is

$$F_L = (0.11)\frac{\pi(0.064 \text{ m})^2}{4}\frac{(1.184 \text{ kg/m}^3)(25.56 \text{ m/s})^2}{2}\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = 0.14 \text{ N}$$

The weight of the ball is $W = mg = (0.057 \text{ kg})(9.81 \text{ m/s}^2)\left(\frac{1 \text{ N}}{1 \text{ kg}\cdot\text{m/s}^2}\right) = 0.56 \text{ N}$

which is more than the lift. Therefore, the ball will **drop** under the combined effect of gravity and lift due to spinning after hitting, with a net force of 0.56 - 0.14 = 0.42 N.

***Discussion*** The Reynolds number for this problem is $\text{Re}_L = \frac{VD}{\nu} = \frac{(25.56 \text{m/s})(0.064 \text{ m})}{1.562\times10^{-5} \text{ m}^2/\text{s}} = 1.05\times10^5$, which is close enough to 6×10$^4$ for which Fig. 11-53 is prepared. Therefore, the result should be close enough to the actual answer.

---

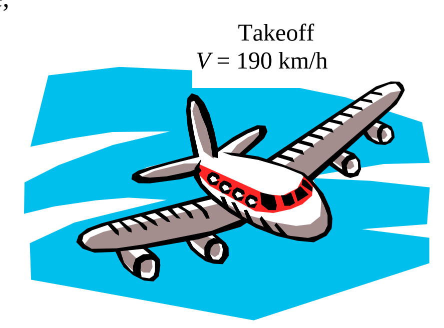
## 11-83

**Solution** The takeoff speed of an aircraft when it is fully loaded is given. The required takeoff speed when the weight of the aircraft is increased by 20% as a result of overloading is to be determined.

***Assumptions*** **1** The atmospheric conditions (and thus the properties of air) remain the same. **2** The settings of the plane during takeoff are maintained the same so that the lift coefficient of the plane remains the same.

***Analysis*** An aircraft will takeoff when lift equals the total weight. Therefore,

$$W = F_L \quad \rightarrow \quad W = \tfrac{1}{2} C_L \rho V^2 A \quad \rightarrow \quad V = \sqrt{\frac{2W}{\rho C_L A}}$$




We note that the takeoff velocity is proportional to the square root of the weight of the aircraft. When the density, lift coefficient, and area remain constant, the ratio of the velocities of the overloaded and fully loaded aircraft becomes

$$\frac{V_2}{V_1} = \frac{\sqrt{2W_2/\rho C_L A}}{\sqrt{2W_1/\rho C_L A}} = \frac{\sqrt{W_2}}{\sqrt{W_1}} \qquad \rightarrow \qquad V_2 = V_1\sqrt{\frac{W_2}{W_1}}$$

Substituting, the takeoff velocity of the overloaded aircraft is determined to be

$$V_2 = V_1\sqrt{\frac{1.2W_1}{W_1}} = (190 \text{ km/h})\sqrt{1.2} = \mathbf{208 \text{ km/h}}$$

***Discussion*** A similar analysis can be performed for the effect of the variations in density, lift coefficient, and planform area on the takeoff velocity.

---

**11-84**

**Solution** The takeoff speed and takeoff time of an aircraft at sea level are given. The required takeoff speed, takeoff time, and the additional runway length required at a higher elevation are to be determined.

***Assumptions*** **1** Standard atmospheric conditions exist. **2** The settings of the plane during takeoff are maintained the same so that the lift coefficient of the plane and the planform area remain constant. **3** The acceleration of the aircraft during takeoff remains constant.

***Properties*** The density of standard air is $\rho_1$ = 1.225 kg/m$^3$ at sea level, and $\rho_2$ = 1.048 kg/m$^3$ at 1600 m altitude.

***Analysis*** (*a*) An aircraft will takeoff when lift equals the total weight. Therefore,

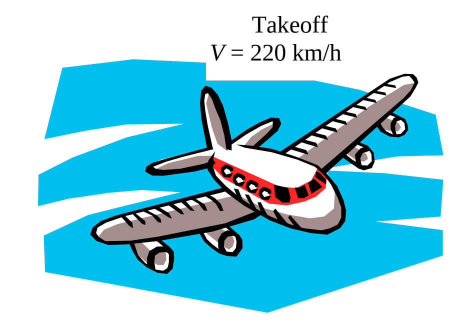
$$W = F_L \quad \rightarrow \quad W = \tfrac{1}{2} C_L \rho V^2 A \quad \rightarrow \quad V = \sqrt{\frac{2W}{\rho C_L A}}$$

We note that the takeoff speed is inversely proportional to the square root of air density. When the weight, lift coefficient, and area remain constant, the ratio of the speeds of the aircraft at high altitude and at sea level becomes

$$\frac{V_2}{V_1} = \frac{\sqrt{2W / \rho_2 C_L A}}{\sqrt{2W / \rho_1 C_L A}} = \frac{\sqrt{\rho_1}}{\sqrt{\rho_2}} \qquad \rightarrow \qquad V_2 = V_1 \sqrt{\frac{\rho_1}{\rho_2}} = (220 \text{ km/h}) \sqrt{\frac{1.225}{1.048}} = \mathbf{238 \text{ km/h}}$$

Therefore, the takeoff velocity of the aircraft at higher altitude is 238 km/h.

(*b*) The acceleration of the aircraft at sea level is



Takeoff
$V$ = 220 km/h

$$a = \frac{\Delta V}{\Delta t} = \frac{220 \text{ km/h - 0}}{15 \text{ s}} \left( \frac{1 \text{ m/s}}{3.6 \text{ km/h}} \right) = 4.074 \text{ m/s}^2$$

which is assumed to be constant both at sea level and the higher altitude. Then the takeoff time at the higher altitude becomes

$$a = \frac{\Delta V}{\Delta t} \quad \rightarrow \quad \Delta t = \frac{\Delta V}{a} = \frac{238 \text{ km/h - 0}}{4.074 \text{ m/s}^2} \left( \frac{1 \text{ m/s}}{3.6 \text{ km/h}} \right) = \mathbf{16.2 \text{ s}}$$

(*c*) The additional runway length is determined by calculating the distance traveled during takeoff for both cases, and taking their difference:

$$L_1 = \tfrac{1}{2} a t_1^2 = \tfrac{1}{2} (4.074 \text{ m/s}^2)(15 \text{ s})^2 = 458 \text{ m}$$

$$L_2 = \tfrac{1}{2} a t_2^2 = \tfrac{1}{2} (4.074 \text{ m/s}^2)(16.2 \text{ s})^2 = 535 \text{ m}$$

$$\Delta L = L_2 - L_1 = 535 - 458 = \mathbf{77 \text{ m}}$$

***Discussion*** Note that altitude has a significant effect on the length of the runways, and it should be a major consideration on the design of airports. It is interesting that a 1.2 second increase in takeoff time increases the required runway length by about 100 m.

**11-85E**

**Solution** The rate of fuel consumption of an aircraft while flying at a low altitude is given. The rate of fuel consumption at a higher altitude is to be determined for the same flight velocity.

***Assumptions*** **1** Standard atmospheric conditions exist. **2** The settings of the plane during takeoff are maintained the same so that the drag coefficient of the plane and the planform area remain constant. **3** The velocity of the aircraft and the propulsive efficiency remain constant. **4** The fuel is used primarily to provide propulsive power to overcome drag, and thus the energy consumed by auxiliary equipment (lights, etc) is negligible.

***Properties*** The density of standard air is $\rho_1$ = 0.05648 lbm/ft$^3$ at 10,000 ft, and $\rho_2$ = 0.02866 lbm/ft$^3$ at 30,000 ft altitude.

***Analysis*** When an aircraft cruises steadily (zero acceleration) at a constant altitude, the net force acting on the aircraft is zero, and thus the thrust provided by the engines must be equal to the drag force. Also, power is force times velocity (distance per unit time), and thus the propulsive power required to overcome drag is equal to the thrust times the cruising velocity. Therefore,

Cruising
$m_{\text{fuel}}$ = 5 gal/min

$$\dot{W}_{\text{propulsive}} = \text{Thrust} \times V = F_D V = C_D A \frac{\rho V^2}{2} V = C_D A \frac{\rho V^3}{2}$$

The propulsive power is also equal to the product of the rate of fuel energy supplied (which is the rate of fuel consumption times the heating value of the fuel, $\dot{m}_{\text{fuel}}\text{HV}$ ) and the propulsive efficiency. Then,

$$\dot{W}_{\text{prop}} = \eta_{\text{prop}} \dot{m}_{\text{fuel}} \text{HV} \quad \rightarrow \quad C_D A \frac{\rho V^3}{2} = \eta_{\text{prop}} \dot{m}_{\text{fuel}} \text{HV}$$

We note that the rate of fuel consumption is proportional to the density of air. When the drag coefficient, the wing area, the velocity, and the propulsive efficiency remain constant, the ratio of the rates of fuel consumptions of the aircraft at high and low altitudes becomes

$$\frac{\dot{m}_{\text{fuel},2}}{\dot{m}_{\text{fuel},1}} = \frac{C_D A \rho_2 V^3 / 2\eta_{\text{prop}}\ \text{HV}}{C_D A \rho_1 V^3 / 2\eta_{\text{prop}}\ \text{HV}} = \frac{\rho_2}{\rho_1} \quad \rightarrow \quad \dot{m}_{\text{fuel},2} = \dot{m}_{\text{fuel},1} \frac{\rho_2}{\rho_1} = (5\ \text{gal/min}) \frac{0.02866}{0.05648} = \mathbf{2.54\ gal/min}$$

***Discussion*** Note the fuel consumption drops by half when the aircraft flies at 30,000 ft instead of 10,000 ft altitude. Therefore, large passenger planes routinely fly at high altitudes (usually between 30,000 and 40,000 ft) to save fuel. This is especially the case for long flights.

**11-86**

**Solution** The takeoff speed of an aircraft when it is fully loaded is given. The required takeoff speed when the aircraft has 100 empty seats is to be determined.

***Assumptions*** **1** The atmospheric conditions (and thus the properties of air) remain the same. **2** The settings of the plane during takeoff are maintained the same so that the lift coefficient of the plane remains the same. **3** A passenger with luggage has an average mass of 140 kg.

Takeoff
$V$ = 250 km/h

***Analysis*** An aircraft will takeoff when lift equals the total weight. Therefore,

$$W = F_L \quad \rightarrow \quad W = \tfrac{1}{2} C_L \rho V^2 A \quad \rightarrow \quad V = \sqrt{\frac{2W}{\rho C_L A}}$$

We note that the takeoff velocity is proportional to the square root of the weight of the aircraft. When the density, lift coefficient, and wing area remain constant, the ratio of the velocities of the under-loaded and fully loaded aircraft becomes

$$\frac{V_2}{V_1} = \frac{\sqrt{2W_2 / \rho C_L A}}{\sqrt{2W_1 / \rho C_L A}} = \frac{\sqrt{W_2}}{\sqrt{W_1}} = \frac{\sqrt{m_2 g}}{\sqrt{m_1 g}} = \frac{\sqrt{m_2}}{\sqrt{m_1}} \quad \rightarrow \quad V_2 = V_1 \sqrt{\frac{m_2}{m_1}}$$

where $m_2 = m_1 - m_{\text{unused capacity}} = 400{,}000\text{ kg} - (140\text{ kg/passanger}) \times (100\text{ passengers}) = 386{,}000\text{ kg}$

Substituting, the takeoff velocity of the overloaded aircraft is determined to be

$$V_2 = V_1 \sqrt{\frac{m_2}{m_1}} = (250\text{ km/h})\sqrt{\frac{386{,}000}{400{,}000}} = \mathbf{246\ km/h}$$

***Discussion*** Note that the effect of empty seats on the takeoff velocity of the aircraft is small. This is because the most weight of the aircraft is due to its empty weight (the aircraft itself rather than the passengers and their luggage.)

---

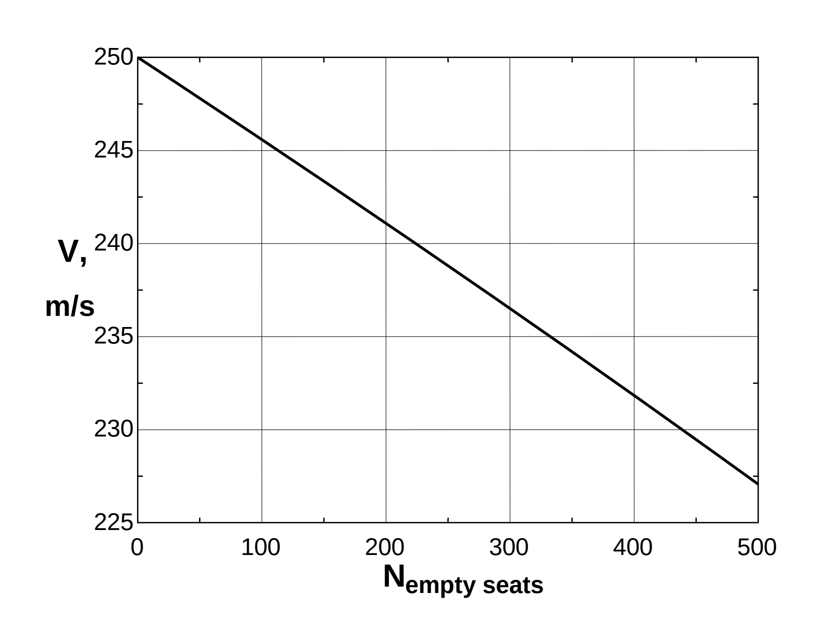
**11-87** 

**Solution** The previous problem is reconsidered. The effect of empty passenger count on the takeoff speed of the aircraft as the number of empty seats varies from 0 to 500 in increments of 50 is to be investigated.

***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
m_passenger=140 "kg"
m1=400000 "kg"
m2=m1-N_empty*m_passenger
V1=250 "km/h"
V2=V1*SQRT(m2/m1)
```

| Empty seats | $m_{\text{airplane,1}}$, kg | $m_{\text{airplane,1}}$, kg | $V_{\text{takeoff}}$, m/s |
|---|---|---|---|
| 0 | 400000 | 400000 | 250.0 |
| 50 | 400000 | 393000 | 247.8 |
| 100 | 400000 | 386000 | 245.6 |
| 150 | 400000 | 379000 | 243.3 |
| 200 | 400000 | 372000 | 241.1 |
| 250 | 400000 | 365000 | 238.8 |
| 300 | 400000 | 358000 | 236.5 |
| 350 | 400000 | 351000 | 234.2 |
| 400 | 400000 | 344000 | 231.8 |
| 450 | 400000 | 337000 | 229.5 |
| 500 | 400000 | 330000 | 227.1 |




***Discussion*** As expected, the takeoff speed decreases as the number of empty seats increases. On the scale plotted, the curve appears nearly linear, but it is not; the curve is actually a small portion of a square-root curve.

---

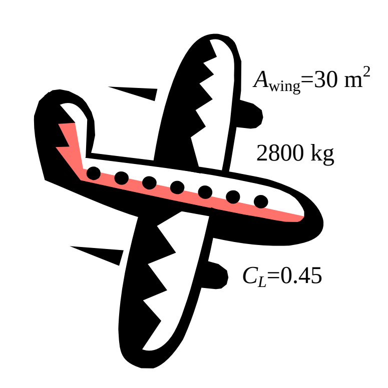
**11-88**

**Solution** The wing area, lift coefficient at takeoff settings, the cruising drag coefficient, and total mass of a small aircraft are given. The takeoff speed, the wing loading, and the required power to maintain a constant cruising speed are to be determined.

***Assumptions*** **1** Standard atmospheric conditions exist. **2** The drag and lift produced by parts of the plane other than the wings are not considered.

***Properties*** The density of standard air at sea level is $\rho$= 1.225 kg/m$^3$.

***Analysis*** (*a*) An aircraft will takeoff when lift equals the total weight. Therefore,




$$W = F_L \quad \rightarrow \quad W = \tfrac{1}{2} C_L \rho V^2 A \quad \rightarrow \quad V = \sqrt{\frac{2W}{\rho C_L A}}$$

Substituting, the takeoff speed is determined to be

$$V_{\text{takeoff}} = \sqrt{\frac{2mg}{\rho C_{L,\text{takeoff}} A}} = \sqrt{\frac{2(2800\text{ kg})(9.81\text{ m/s}^2)}{(1.225\text{ kg/m}^3)(0.45)(30\text{ m}^2)}}$$
$$= 57.6\text{ m/s} = \mathbf{207\ km/h}$$

(*b*) Wing loading is the average lift per unit planform area, which is equivalent to the ratio of the lift to the planform area of the wings since the lift generated during steady cruising is equal to the weight of the aircraft. Therefore,

$$F_{\text{loading}} = \frac{F_L}{A} = \frac{W}{A} = \frac{(2800\text{ kg})(9.81\text{ m/s}^2)}{30\text{ m}^2} = \mathbf{916\ N/m^2}$$

(*c*) When the aircraft is cruising steadily at a constant altitude, the net force acting on the aircraft is zero, and thus thrust provided by the engines must be equal to the drag force, which is

$$F_D = C_D A \frac{\rho V^2}{2} = (0.035)(30\text{ m}^2)\frac{(1.225\text{ kg/m}^3)(300/3.6\text{ m/s})^2}{2}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 4.466\text{ kN}$$

Noting that power is force times velocity, the propulsive power required to overcome this drag is equal to the thrust times the cruising velocity,

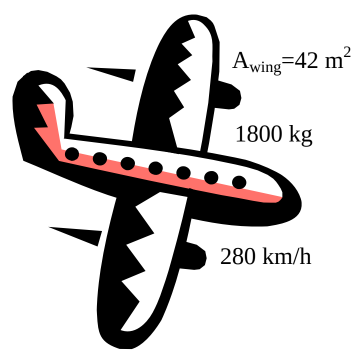
$$\text{Power} = \text{Thrust} \times \text{Velocity} = F_D V = (4.466\text{ kN})(300/3.6\text{ m/s})\left(\frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}}\right) = \mathbf{372\ kW}$$

Therefore, the engines must supply 372 kW of propulsive power to overcome the drag during cruising.

***Discussion*** The power determined above is the power to overcome the drag that acts on the wings only, and does not include the drag that acts on the remaining parts of the aircraft (the fuselage, the tail, etc). Therefore, the total power required during cruising will be greater. The required rate of energy input can be determined by dividing the propulsive power by the propulsive efficiency.

**11-89**

**Solution** The total mass, wing area, cruising speed, and propulsive power of a small aircraft are given. The lift and drag coefficients of this airplane while cruising are to be determined.




***Assumptions*** **1** Standard atmospheric conditions exist. **2** The drag and lift produced by parts of the plane other than the wings are not considered. **3** The fuel is used primarily to provide propulsive power to overcome drag, and thus the energy consumed by auxiliary equipment (lights, etc) is negligible.

***Properties*** The density of standard air at an altitude of 4000 m is $\rho$= 0.819 kg/m$^3$.

***Analysis*** Noting that power is force times velocity, the propulsive power required to overcome this drag is equal to the thrust times the cruising velocity. Also, when the aircraft is cruising steadily at a constant altitude, the net force acting on the aircraft is zero, and thus thrust provided by the engines must be equal to the drag force. Then,

$$\dot{W}_{\text{prop}} = \text{Thrust} \times \text{Velocity} = F_D V \quad \rightarrow \quad F_D = \frac{\dot{W}_{\text{prop}}}{V} = \frac{190 \text{ kW}}{280/3.6 \text{ m/s}}\left(\frac{1000 \text{ N}\cdot\text{m/s}}{1 \text{ kW}}\right) = 2443 \text{ N}$$

Then the drag coefficient becomes

$$F_D = C_D A \frac{\rho V^2}{2} \quad \rightarrow \quad C_D = \frac{2F_D}{\rho A V^2} = \frac{2(2443 \text{ N})}{(0.819 \text{ kg/m}^3)(42 \text{ m}^2)(280/3.6 \text{ m/s})^2}\left(\frac{1 \text{ kg}\cdot\text{m/s}^2}{1 \text{ N}}\right) = \mathbf{0.0235}$$

An aircraft cruises at constant altitude when lift equals the total weight. Therefore,

$$W = F_L = \tfrac{1}{2} C_L \rho V^2 A \quad \rightarrow \quad C_L = \frac{2W}{\rho V^2 A} = \frac{2(1800 \text{ kg})(9.81 \text{ m/s}^2)}{(0.819 \text{ kg/m}^3)(42 \text{ m}^2)(280/3.6 \text{ m/s})^2} = \mathbf{0.17}$$

Therefore, the drag and lift coefficients of this aircraft during cruising are 0.0235 and 0.17, respectively, with a $C_L/C_D$ ratio of 7.2.

***Discussion*** The drag and lift coefficient determined are for cruising conditions. The values of these coefficient can be very different during takeoff because of the angle of attack and the wing geometry.

---

**11-90**

**Solution** An airfoil has a given lift-to drag ratio at 0° angle of attack. The angle of attack that will raise this ratio to 80 is to be determined.

***Analysis*** The ratio $C_L/C_D$ for the given airfoil is plotted against the angle of attack in Fig. 11-43. The angle of attack corresponding to $C_L/C_D$ = 80 is $\theta$ = **3°**.

***Discussion*** Note that different airfoils have different $C_L/C_D$ vs. $\theta$ charts.

---

**11-91**

**Solution** The wings of a light plane resemble the NACA 23012 airfoil with no flaps. Using data for that airfoil, the takeoff speed at a specified angle of attack and the stall speed are to be determined.

***Assumptions*** **1** Standard atmospheric conditions exist. **2** The drag and lift produced by parts of the plane other than the wings are not considered.

***Properties*** The density of standard air at sea level is $\rho$ = 1.225 kg/m$^3$. At an angle of attack of 5°, the lift and drag coefficients are read from Fig. 11-45 to be $C_L$ = 0.6 and $C_D$ = 0.015. The maximum lift coefficient is $C_{L,\max}$ = 1.52 and it occurs at an angle of attack of 15°.

***Analysis*** An aircraft will takeoff when lift equals the total weight. Therefore,

$$W = F_L \quad \rightarrow \quad W = \tfrac{1}{2} C_L \rho V^2 A \quad \rightarrow \quad V = \sqrt{\frac{2W}{\rho C_L A}}$$

$A_{wing}$ = 46 m$^2$

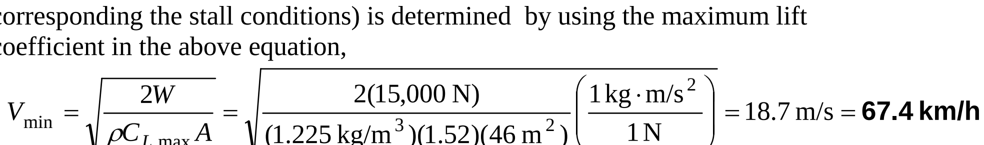
$W$ = 15,000 N

Substituting, the takeoff speed is determined to be

$$V_{\text{takeoff}} = \sqrt{\frac{2(15{,}000\text{ N})}{(1.225\text{ kg/m}^3)(0.6)(46\text{ m}^2)}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)} = 29.8\text{ m/s} = \mathbf{107\ km/h}$$

since 1 m/s = 3.6 km/h. The stall velocity (the minimum takeoff velocity corresponding the stall conditions) is determined by using the maximum lift coefficient in the above equation,





$$V_{\min} = \sqrt{\frac{2W}{\rho C_{L,\max} A}} = \sqrt{\frac{2(15{,}000\text{ N})}{(1.225\text{ kg/m}^3)(1.52)(46\text{ m}^2)}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)} = 18.7\text{ m/s} = \mathbf{67.4\ km/h}$$

***Discussion*** The "safe" minimum velocity to avoid the stall region is obtained by multiplying the stall velocity by 1.2:

$$V_{\text{min,safe}} = 1.2V_{\min} = 1.2\times(18.7\text{ m/s}) = 22.4\text{ m/s} = 80.8\text{ km/h}$$

Note that the takeoff velocity decreased from 107 km/h at an angle of attack of 5° to 80.8 km/s under stall conditions with a safety margin.

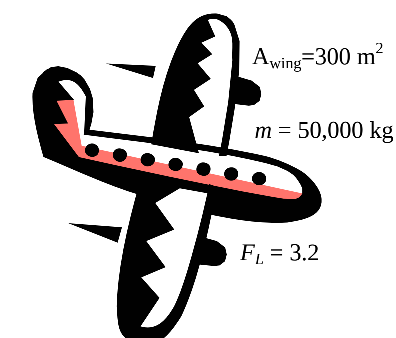
**11-92** **[***Also solved using EES on enclosed DVD***]**

**Solution** The mass, wing area, the maximum (stall) lift coefficient, the cruising speed and the cruising drag coefficient of an airplane are given. The safe takeoff speed at sea level and the thrust that the engines must deliver during cruising are to be determined.

***Assumptions*** **1** Standard atmospheric conditions exist **2** The drag and lift produced by parts of the plane other than the wings are not considered. **3** The takeoff speed is 20% over the stall speed. **4** The fuel is used primarily to provide propulsive power to overcome drag, and thus the energy consumed by auxiliary equipment (lights, etc) is negligible.

***Properties*** The density of standard air is $\rho_1$ = 1.225 kg/m$^3$ at sea level, and $\rho_2$ = 0.312 kg/m$^3$ at 12,000 m altitude. The cruising drag coefficient is given to be $C_D$ = 0.03. The maximum lift coefficient is given to be $C_{L,\max}$ = 3.2.

***Analysis*** (*a*) An aircraft will takeoff when lift equals the total weight. Therefore,



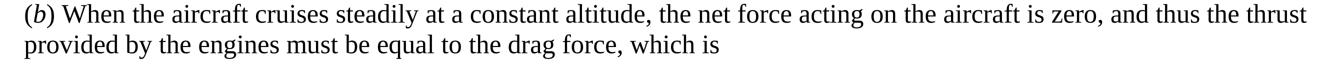

$$W = F_L \quad \rightarrow \quad W = \tfrac{1}{2} C_L \rho V^2 A \quad \rightarrow \quad V = \sqrt{\frac{2W}{\rho C_L A}} = \sqrt{\frac{2mg}{\rho C_L A}}$$

The stall velocity (the minimum takeoff velocity corresponding the stall conditions) is determined by using the maximum lift coefficient in the above equation,

$$V_{\min} = \sqrt{\frac{2mg}{\rho_1 C_{L,\max} A}} = \sqrt{\frac{2(50{,}000\text{ kg})(9.81\text{ m/s}^2)}{(1.225\text{ kg/m}^3)(3.2)(300\text{ m}^2)}} = 28.9\text{ m/s} = 104\text{ km/h}$$

since 1 m/s = 3.6 km/h. Then the “safe” minimum velocity to avoid the stall region becomes

$$V_{\min,\text{safe}} = 1.2 V_{\min} = 1.2 \times (28.9\text{ m/s}) = 34.7\text{ m/s} = \mathbf{125\text{ km/h}}$$



(*b*) When the aircraft cruises steadily at a constant altitude, the net force acting on the aircraft is zero, and thus the thrust provided by the engines must be equal to the drag force, which is

$$F_D = C_D A \frac{\rho_2 V^2}{2} = (0.03)(300\text{ m}^2)\frac{(0.312\text{ kg/m}^3)(700/3.6\text{ m/s})^2}{2}\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 53.08\text{ kN}$$

Noting that power is force times velocity, the propulsive power required to overcome this drag is equal to the thrust times the cruising velocity,

$$\text{Power} = \text{Ttrust} \times \text{Velocity} = F_D V = (53.08\text{ kN})(700/3.6\text{ m/s})\left(\frac{1\text{ kW}}{1\text{ kN}\cdot\text{m/s}}\right) = \mathbf{10{,}300\text{ kW}}$$

Therefore, the engines must supply 10,300 kW of propulsive power to overcome drag during cruising.

***Discussion*** The power determined above is the power to overcome the drag that acts on the wings only, and does not include the drag that act on the remaining parts of the aircraft (the fuselage, the tail, etc). Therefore, the total power required during cruising will be greater. The required rate of energy input can be determined by dividing the propulsive power by the propulsive efficiency.

**11-93E**

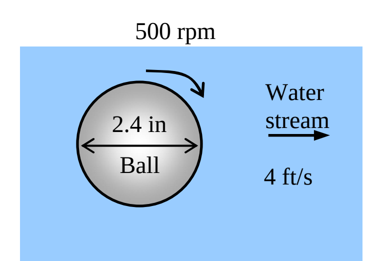
**Solution** A spinning ball is dropped into a water stream. The lift and drag forces acting on the ball are to be determined.

***Assumptions*** **1** The outer surface of the ball is smooth enough for Fig. 11-53 to be applicable. **2** The ball is completely immersed in water.

***Properties*** The density and dynamic viscosity of water at 60°F are $\rho$ = 62.36 lbm/ft$^3$ and $\mu$ = 2.713 lbm/ft·h = 7.536×10$^{-4}$ lbm/ft·s.

***Analysis*** The drag and lift forces can be determined from

$$F_D = C_D A \frac{\rho V^2}{2} \quad \text{and} \quad F_L = C_L A \frac{\rho V^2}{2}$$




where $A$ is the frontal area of the ball, which is $A = \pi D^2/4$, and $D$ = 2.4/12 = 0.2 ft. The Reynolds number and the angular velocity of the ball are

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(62.36\ \text{lbm/ft}^3)(4\ \text{ft/s})(0.2\ \text{ft})}{7.536\times10^{-4}\ \text{ft}^2/\text{s}} = 6.62\times10^4$$

$$\omega = (500\ \text{rev/min})\left(\frac{2\pi\ \text{rad}}{1\ \text{rev}}\right)\left(\frac{1\ \text{min}}{60\ \text{s}}\right) = 52.4\ \text{rad/s}$$

and

$$\frac{\omega D}{2V} = \frac{(52.4\ \text{rad/s})(0.2\ \text{ft})}{2(4\ \text{ft/s})} = 1.31\ \text{rad}$$

From Fig. 11-53, the drag and lift coefficients corresponding to this value are $C_D$ = 0.56 and $C_L$ = 0.35. Then the drag and the lift acting on the ball are

$$F_D = (0.56)\frac{\pi(0.2\ \text{ft})^2}{4}\frac{(62.36\ \text{lbm/ft}^3)(4\ \text{ft/s})^2}{2}\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right) = \mathbf{0.27\ lbf}$$

$$F_L = (0.35)\frac{\pi(0.2\ \text{ft})^2}{4}\frac{(62.36\ \text{lbm/ft}^3)(4\ \text{ft/s})^2}{2}\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right) = \mathbf{0.17\ lbf}$$

***Discussion*** The Reynolds number for this problem is 6.62×10$^4$ which is close enough to 6×10$^4$ for which Fig. 11-53 is prepared. Therefore, the result should be close enough to the actual answer.

## Review Problems

**11-94**

**Solution** An automotive engine is approximated as a rectangular block. The drag force acting on the bottom surface of the engine is to be determined.

***Assumptions*** **1** The air flow is steady and incompressible. **2** Air is an ideal gas. **3** The atmospheric air is calm (no significant winds). **3** The air flow is turbulent over the entire surface because of the constant agitation of the engine block. **4** The bottom surface of the engine is a flat surface, and it is smooth (in reality it is quite rough because of the dirt collected on it).

***Properties*** The density and kinematic viscosity of air at 1 atm and 15°C are $\rho$ = 1.225 kg/m$^3$ and $\nu$ = 1.470×10$^{-5}$ m$^2$/s.

***Analysis*** The Reynolds number at the end of the engine block is

$$\mathrm{Re}_L = \frac{VL}{\nu} = \frac{\left[(85/3.6)\ \mathrm{m/s}\right](0.7\ \mathrm{m})}{1.470\times10^{-5}\ \mathrm{m^2/s}} = 1.124\times10^6$$

Engine block

Air
$V$ = 85 km/h
$T$ = 15°C

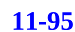
$L$ = 0.7 m

The flow is assumed to be turbulent over the entire surface. Then the average friction coefficient and the drag force acting on the surface becomes

$$C_f = \frac{0.074}{\mathrm{Re}_L^{1/5}} = \frac{0.074}{(1.124\times10^6)^{1/5}} = 0.004561$$

$$F_D = C_f A\frac{\rho V^2}{2} = (0.004561)[0.6\times0.7\ \mathrm{m^2}]\frac{(1.225\ \mathrm{kg/m^3})(85/3.6\ \mathrm{m/s})^2}{2}\left(\frac{1\ \mathrm{N}}{1\ \mathrm{kg.m/s^2}}\right) = \mathbf{0.65\ N}$$

***Discussion*** Note that the calculated drag force (and the power required to overcome it) is very small. This is not surprising since the drag force for blunt bodies is almost entirely due to pressure drag, and the friction drag is practically negligible compared to the pressure drag.



**11-95** 

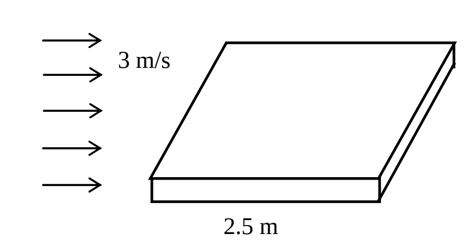
**Solution** A fluid flows over a 2.5-m long flat plate. The thickness of the boundary layer at intervals of 0.25 m is to be determined and plotted against the distance from the leading edge for air, water, and oil.

***Assumptions*** **1** The flow is steady and incompressible. **2** The critical Reynolds number is $Re_{cr} = 5\times10^5$. **3** Air is an ideal gas. **4** The surface of the plate is smooth.

***Properties*** The kinematic viscosity of the three fluids at 1 atm and 20°C are: $\nu = 1.516\times10^{-5}$ m²/s for air , $\nu = \mu/\rho = (1.002\times10^{-3}\text{ kg/m·s})/(998\text{ kg/m}^3) = 1.004\times10^{-6}$ m²/s for water, and $\nu = 9.429\times10^{-4}$ m²/s for oil.

***Analysis*** The thickness of the boundary layer along the flow for laminar and turbulent flows is given by

Laminar flow: $\delta_x = \dfrac{4.91x}{Re_x^{1/2}}$, Turbulent flow: $\delta_x = \dfrac{0.38x}{Re_x^{1/5}}$

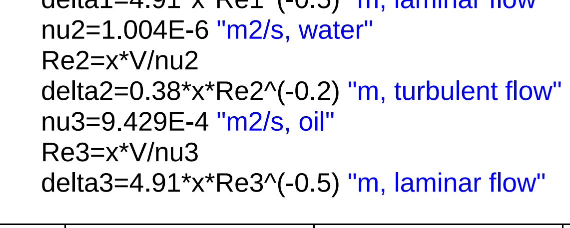



(*a*) AIR: The Reynolds number and the boundary layer thickness at the end of the first 0.25 m interval are

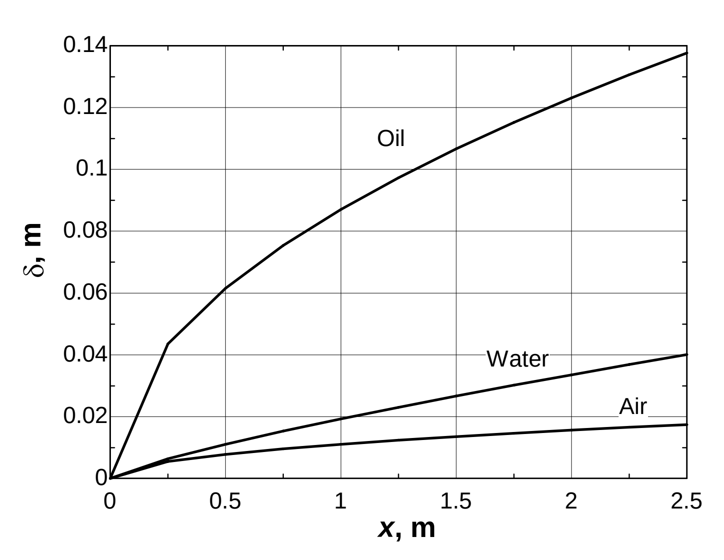
$$Re_x = \frac{Vx}{\nu} = \frac{(3\text{ m/s})(0.25\text{ m})}{1.516\times10^{-5}\text{ m}^2\text{/s}} = 0.495\times10^5,$$

$$\delta_x = \frac{5x}{Re_x^{1/2}} = \frac{4.91\times(0.25\text{ m})}{(0.495\times10^5)^{0.5}} = 5.52\times10^{-3}\text{ m}$$

We repeat calculations for all 0.25 m intervals. The EES *Equations* window is printed below, along with the tabulated and plotted results.



```
V=3 "m/s"
nu1=1.516E-5 "m2/s, Air"
Re1=x*V/nu1
delta1=4.91*x*Re1^(-0.5) "m, laminar flow"
nu2=1.004E-6 "m2/s, water"
Re2=x*V/nu2
delta2=0.38*x*Re2^(-0.2) "m, turbulent flow"
nu3=9.429E-4 "m2/s, oil"
Re3=x*V/nu3
delta3=4.91*x*Re3^(-0.5) "m, laminar flow"
```




| *x*, cm | Air | | Water | | Oil | |
|---|---|---|---|---|---|---|
| | Re | $\delta_x$ | Re | $\delta_x$ | Re | $\delta_x$ |
| 0.00 | 0.000E+00 | 0.0000 | 0.000E+00 | 0.0000 | 0.000E+00 | 0.0000 |
| 0.25 | 4.947E+04 | 0.0055 | 7.470E+05 | 0.0064 | 7.954E+02 | 0.0435 |
| 0.50 | 9.894E+04 | 0.0078 | 1.494E+06 | 0.0111 | 1.591E+03 | 0.0616 |
| 0.75 | 1.484E+05 | 0.0096 | 2.241E+06 | 0.0153 | 2.386E+03 | 0.0754 |
| 1.00 | 1.979E+05 | 0.0110 | 2.988E+06 | 0.0193 | 3.182E+03 | 0.0870 |
| 1.25 | 2.474E+05 | 0.0123 | 3.735E+06 | 0.0230 | 3.977E+03 | 0.0973 |
| 1.50 | 2.968E+05 | 0.0135 | 4.482E+06 | 0.0266 | 4.773E+03 | 0.1066 |
| 1.75 | 3.463E+05 | 0.0146 | 5.229E+06 | 0.0301 | 5.568E+03 | 0.1152 |
| 2.00 | 3.958E+05 | 0.0156 | 5.976E+06 | 0.0335 | 6.363E+03 | 0.1231 |
| 2.25 | 4.453E+05 | 0.0166 | 6.723E+06 | 0.0369 | 7.159E+03 | 0.1306 |
| 2.50 | 4.947E+05 | 0.0175 | 7.470E+06 | 0.0401 | 7.954E+03 | 0.1376 |

***Discussion*** Note that the flow is laminar for (a) and (c), and turbulent for (b). Also note that the thickness of the boundary layer is very small for air and water, but it is very large for oil. This is due to the high viscosity of oil.

## 11-96E

**Solution** The passenger compartment of a minivan is modeled as a rectangular box. The drag force acting on the top and the two side surfaces and the power needed to overcome it are to be determined.

***Assumptions*** **1** The air flow is steady and incompressible. **2** The air flow over the exterior surfaces is turbulent because of constant agitation. **3** Air is an ideal gas. **4** The top and side surfaces of the minivan are flat and smooth (in reality they can be rough). **5** The atmospheric air is calm (no significant winds).

***Properties*** The density and kinematic viscosity of air at 1 atm and 80°F are $\rho = 0.07350$ lbm/ft$^3$ and $\nu = 0.6110$ ft$^2$/h = $1.697\times10^{-4}$ ft$^2$/s.

***Analysis*** The Reynolds number at the end of the top and side surfaces is

$$\mathrm{Re}_L = \frac{VL}{\nu} = \frac{\left[(60\times1.4667)\ \text{ft/s}\right](11\ \text{ft})}{1.697\times10^{-4}\ \text{ft}^2/\text{s}} = 5.704\times10^6$$

The air flow over the entire outer surface is assumed to be turbulent. Then the friction coefficient becomes

$$C_f = \frac{0.074}{\mathrm{Re}_L^{1/5}} = \frac{0.074}{(5.704\times10^6)^{1/5}} = 0.00330$$

The area of the top and side surfaces of the minivan is

$$A = A_{\text{top}} + 2A_{\text{side}} = 6\times11+2\times3.2\times11 = 136.4\ \text{ft}^2$$

Noting that the pressure drag is zero and thus $C_D = C_f$ for a plane surface, the drag force acting on these surfaces becomes

$$F_D = C_f A\frac{\rho V^2}{2} = 0.00330\times(136.4\ \text{ft}^2)\frac{(0.074\ \text{lbm/ft}^3)(60\times1.4667\ \text{ft/s})^2}{2}\left(\frac{1\ \text{lbf}}{32.2\ \text{lbm}\cdot\text{ft/s}^2}\right) = \mathbf{4.0\ lbf}$$

Noting that power is force times velocity, the power needed to overcome this drag force is

$$\dot{W}_{\text{drag}} = F_D V = (4.0\ \text{lbf})(60\times1.4667\ \text{ft/s})\left(\frac{1\ \text{kW}}{737.56\ \text{lbf}\cdot\text{ft/s}}\right) = \mathbf{0.48\ kW}$$

***Discussion*** Note that the calculated drag force (and the power required to overcome it) is very small. This is not surprising since the drag force for blunt bodies is almost entirely due to pressure drag, and the friction drag is practically negligible compared to the pressure drag.

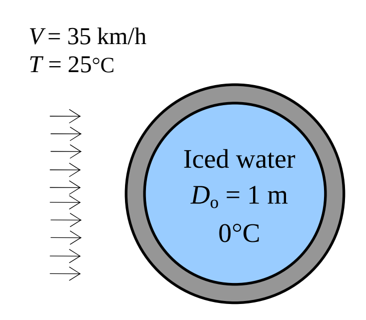
---

## 11-97

**Solution** A large spherical tank located outdoors is subjected to winds. The drag force exerted on the tank by the winds is to be determined.

***Assumptions*** **1** The outer surfaces of the tank are smooth. **2** Air flow in the wind is steady and incompressible, and flow around the tank is uniform. **3** Turbulence in the wind is not considered. **4** The effect of any support bars on flow and drag is negligible.

***Properties*** The density and kinematic viscosity of air at 1 atm and 25°C are $\rho$ = 1.184 kg/m$^3$ and $\nu = 1.562\times10^{-5}$ m$^2$/s.




***Analysis*** Noting that $D$ = 1 m and 1 m/s = 3.6 km/h, the Reynolds number for the flow is

$$\mathrm{Re} = \frac{VD}{\nu} = \frac{\left[(35/3.6)\ \text{m/s}\right](1\ \text{m})}{1.562\times10^{-5}\ \text{m}^2/\text{s}} = 6.224\times10^5$$

The drag coefficient for a smooth sphere corresponding to this Reynolds number is, from Fig. 11-36, $C_D$ = 0.065. Also, the frontal area for flow past a sphere is $A = \pi D^2/4$. Then the drag force becomes

$$F_D = C_D A\frac{\rho V^2}{2} = 0.065\left[\pi\left(1\ \text{m}^2\right)/4\right]\frac{\left(1.184\ \text{kg/m}^3\right)\left(35/3.6\ \text{m/s}\right)^2}{2}\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = \mathbf{2.86\ N}$$

***Discussion*** Note that the drag coefficient is very low in this case since the flow is turbulent (Re > $2\times10^5$). Also, it should be kept in mind that wind turbulence may affect the drag coefficient.

---

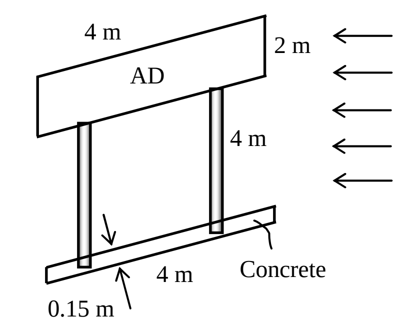
**11-98**

**Solution** A rectangular advertisement panel attached to a rectangular concrete block by two poles is to withstand high winds. For a given maximum wind speed, the maximum drag force on the panel and the poles, and the minimum length *L* of the concrete block for the panel to resist the winds are to be determined.

***Assumptions*** **1** The flow of air is steady and incompressible. **2** The wind is normal to the panel (to check for the worst case). **3** The flow is turbulent so that the tabulated value of the drag coefficients can be used.

***Properties*** In turbulent flow, the drag coefficient is $C_D$ = 0.3 for a circular rod, and $C_D$ = 2.0 for a thin rectangular plate (Table 11-2). The densities of air and concrete block are given to be $\rho$ = 1.30 kg/m$^3$ and $\rho_c$ = 2300 kg/m$^3$.

***Analysis*** (*a*) The drag force acting on the panel is




$$F_{D,\text{panel}} = C_D A \frac{\rho V^2}{2}$$
$$= (2.0)(2\times 4\ \text{m}^2)\frac{(1.30\ \text{kg/m}^3)(150/3.6\ \text{m/s})^2}{2}\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)$$
$$= \mathbf{18{,}000\ N}$$

(*b*) The drag force acting on each pole is

$$F_{D,\text{pole}} = C_D A \frac{\rho V^2}{2}$$
$$= (0.3)(0.05\times 4\ \text{m}^2)\frac{(1.30\ \text{kg/m}^3)(150/3.6\ \text{m/s})^2}{2}\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right)$$
$$= \mathbf{68\ N}$$

Therefore, the drag force acting on both poles is 68 × 2 = **136 N**. Note that the drag force acting on poles is negligible compared to the drag force acting on the panel.

(*c*) The weight of the concrete block is

$$W = mg = \rho g V = (2300\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(\text{L}\times 4\ \text{m}\times 0.15\ \text{m})\left(\frac{1\ \text{N}}{1\ \text{kg}\cdot\text{m/s}^2}\right) = 13{,}540L\ \ \text{N}$$

Note that the resultant drag force on the panel passes through its center, the drag force on the pole passes through the center of the pole, and the weight of the panel passes through the center of the block. When the concrete block is first tipped, the wind-loaded side of the block will be lifted off the ground, and thus the entire reaction force from the ground will act on the other side. Taking the moment about this side and setting it equal to zero gives

$$\sum M = 0 \ \rightarrow\ F_{D,\text{panel}}\times(1+4+0.15) + F_{D,\text{pole}}\times(2+0.15) - W\times(L/2) = 0$$

Substituting and solving for *L* gives

$$18{,}000\times 5.15 + 136\times 2.15 - 13{,}540L\times L/2 = 0 \ \rightarrow\ \text{L} = 3.70\ \text{m}$$

Therefore, the minimum length of the concrete block must be *L* = **3.70**.

***Discussion*** This length appears to be large and impractical. It can be reduced to a more reasonable value by (*a*) increasing the height of the concrete block, (*b*) reducing the length of the poles (and thus the tipping moment), or (*c*) by attaching the concrete block to the ground (through long nails, for example).

**11-99**

**Solution** The bottom surface of a plastic boat is approximated as a flat surface. The friction drag exerted on the bottom surface of the boat by water and the power needed to overcome it are to be determined.

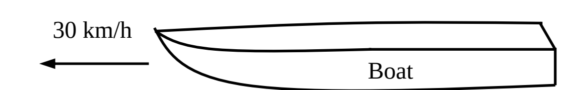
***Assumptions*** **1** The flow is steady and incompressible. **2** The water is calm (no significant currents or waves). **3** The water flow is turbulent over the entire surface because of the constant agitation of the boat. **4** The bottom surface of the boat is a flat surface, and it is smooth.

***Properties*** The density and dynamic viscosity of water at 15°C are $\rho$ = 999.1 kg/m$^3$ and $\mu$ = 1.138×10$^{-3}$ kg/m·s.

***Analysis*** The Reynolds number at the end of the bottom surface of the boat is

$$\text{Re}_L = \frac{\rho VL}{\mu} = \frac{(999.1\ \text{kg/m}^3)(30/3.6\ \text{m/s})(2\ \text{m})}{1.138\times10^{-3}\ \text{kg/m}\cdot\text{s}} = 1.463\times10^7$$

The flow is assumed to be turbulent over the entire surface. Then the average friction coefficient and the drag force acting on the surface becomes




$$C_f = \frac{0.074}{\text{Re}_L^{1/5}} = \frac{0.074}{(1.463\times10^7)^{1/5}} = 0.00273$$

$$F_D = C_f A\frac{\rho V^2}{2} = (0.00273)[1.5\times 2\ \text{m}^2]\frac{(999.1\ \text{kg/m}^3)(30/3.6\ \text{m/s})^2}{2}\left(\frac{1\ \text{N}}{1\ \text{kg.m/s}^2}\right) = \mathbf{284.1N}$$

Noting that power is force times velocity, the power needed to overcome this drag force is

$$\dot{W}_{\text{drag}} = F_D V = (284.1\ \text{N})(30/3.6\ \text{m/s})\left(\frac{1\ \text{kW}}{1000\ \text{N}\cdot\text{m/s}}\right) = \mathbf{2.37\ \ kW}$$

***Discussion*** Note that the calculated drag force (and the power required to overcome it) is relatively small. This is not surprising since the drag force for blunt bodies (including those partially immersed in a liquid) is almost entirely due to pressure drag, and the friction drag is practically negligible compared to the pressure drag.


**11-100** 

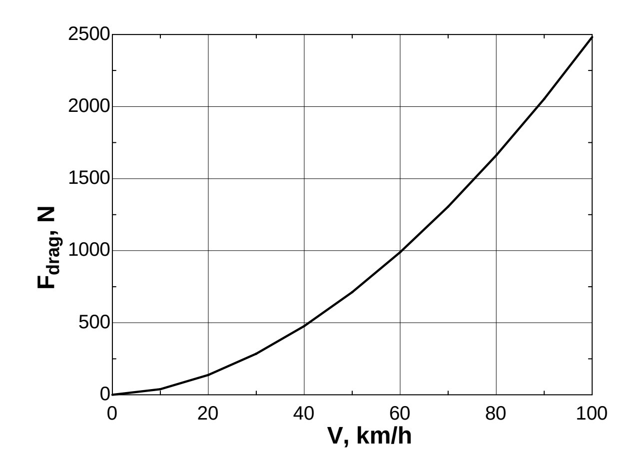
**Solution** The previous problem is reconsidered. The effect of boat speed on the drag force acting on the bottom surface of the boat and the power needed to overcome as the boat speed varies from 0 to 100 km/h in increments of 10 km/h is to be investigated.

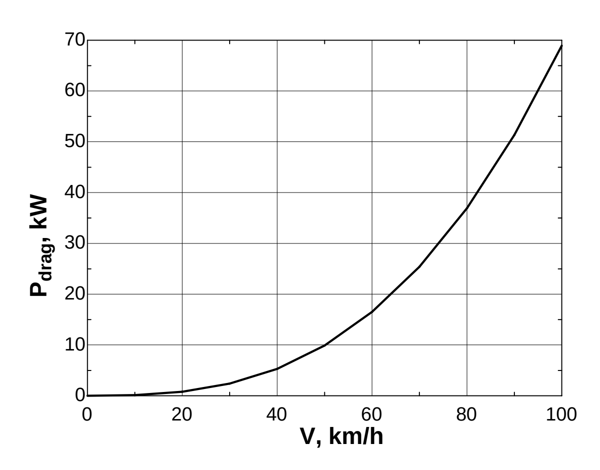
***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
rho=999.1 "kg/m3"
mu=1.138E-3 "m2/s"
V=Vel/3.6 "m/s"
L=2 "m"
W=1.5 "m"
A=L*W

Re=rho*L*V/mu
Cf=0.074/Re^0.2
g=9.81 "m/s2"
F=Cf*A*(rho*V^2)/2 "N"
P_drag=F*V/1000 "kW"
```

| $V$, km/h | Re | $C_f$ | $F_{drag}$, $N$ | $P_{\text{drag},}$ kW |
|---|---|---|---|---|
| 0 | 0 | 0 | 0 | 0.0 |
| 10 | 4.877E+06 | 0.00340 | 39 | 0.1 |
| 20 | 9.755E+06 | 0.00296 | 137 | 0.8 |
| 30 | 1.463E+07 | 0.00273 | 284 | 2.4 |
| 40 | 1.951E+07 | 0.00258 | 477 | 5.3 |
| 50 | 2.439E+07 | 0.00246 | 713 | 9.9 |
| 60 | 2.926E+07 | 0.00238 | 989 | 16.5 |
| 70 | 3.414E+07 | 0.00230 | 1306 | 25.4 |
| 80 | 3.902E+07 | 0.00224 | 1661 | 36.9 |
| 90 | 4.390E+07 | 0.00219 | 2053 | 51.3 |
| 100 | 4.877E+07 | 0.00215 | 2481 | 68.9 |




***Discussion*** The curves look similar at first glance, but in fact $F_{\text{drag}}$ increases like $V^2$, while $F_{\text{drag}}$ increases like $V^3$.

**11-101E** **[*Also solved using EES on enclosed DVD*]**

**Solution** Cruising conditions of a passenger plane are given. The minimum safe landing and takeoff speeds with and without flaps, the angle of attack during cruising, and the power required are to be determined

***Assumptions*** **1** The drag and lift produced by parts of the plane other than the wings are not considered. **2** The wings are assumed to be two-dimensional airfoil sections, and the tip effects are neglected. **4** The lift and drag characteristics of the wings can be approximated by NACA 23012 so that Fig. 11-45 is applicable.

***Properties*** The densities of air are 0.075 lbm/ft$^3$ on the ground and 0.0208 lbm/ft$^3$ at cruising altitude. The maximum lift coefficients of the wings are 3.48 and 1.52 with and without flaps, respectively (Fig. 11-45).



$V$ = 550 mph
$m$ = 150,000 lbm
$A_{wing}$ = 1800 m$^2$

***Analysis*** (*a*) The weight and cruising speed of the airplane are

$$W = mg = (150{,}000\text{ lbm})(32.2\text{ ft/s}^2)\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = 150{,}000\text{ lbf}$$

$$V = (550\text{ mph})\left(\frac{1.4667\text{ ft/s}}{1\text{ mph}}\right) = 806.7\text{ ft/s}$$

Minimum velocity corresponding the stall conditions with and without flaps are

$$V_{\min 1} = \sqrt{\frac{2W}{\rho C_{L,\max 1} A}} = \sqrt{\frac{2(150{,}000\text{ lbf})}{(0.075\text{ lbm/ft}^3)(1.52)(1800\text{ ft}^2)}\left(\frac{32.2\text{ lbm}\cdot\text{ft/s}^2}{1\text{ lbf}}\right)} = 217\text{ ft/s}$$

$$V_{\min 2} = \sqrt{\frac{2W}{\rho C_{L,\max 2} A}} == \sqrt{\frac{2(150{,}000\text{ lbf})}{(0.075\text{ lbm/ft}^3)(3.48)(1800\text{ ft}^2)}\left(\frac{32.2\text{ lbm}\cdot\text{ft/s}^2}{1\text{ lbf}}\right)} = 143\text{ ft/s}$$

The "safe" minimum velocities to avoid the stall region are obtained by multiplying these values by 1.2:

*Without flaps*: $V_{\min 1,\text{safe}} = 1.2V_{\min 1} = 1.2\times(217\text{ ft/s}) = 260\text{ ft/s} = \mathbf{178\text{ mph}}$

*With flaps*: $V_{\min 2,\text{safe}} = 1.2V_{\min 2} = 1.2\times(143\text{ ft/s}) = 172\text{ ft/s} = \mathbf{117\text{ mph}}$

since 1 mph = 1.4667 ft/s. Note that the use of flaps allows the plane to takeoff and land at considerably lower velocities, and thus at a shorter runway.

(*b*) When an aircraft is cruising steadily at a constant altitude, the lift must be equal to the weight of the aircraft, $F_L = W$. Then the lift coefficient is determined to be

$$C_L = \frac{F_L}{\frac{1}{2}\rho V^2 A} = \frac{150{,}000\text{ lbf}}{\frac{1}{2}(0.0208\text{ lbm/ft}^3)(806.7\text{ ft/s})^2(1800\text{ ft}^2)}\left(\frac{32.2\text{ lbm}\cdot\text{ft/s}^2}{1\text{ lbf}}\right) = 0.40$$

For the case of no flaps, the angle of attack corresponding to this value of $C_L$ is determined from Fig. 11-45 to be about α = **3.5°**.

(*c*) When aircraft cruises steadily, the net force acting on the aircraft is zero, and thus thrust provided by the engines must be equal to the drag force. The drag coefficient corresponding to the cruising lift coefficient of 0.40 is $C_D$ = 0.015 (Fig. 11-45). Then the drag force acting on the wings becomes

$$F_D = C_D A\frac{\rho V^2}{2} = (0.015)(1800\text{ ft}^2)\frac{(0.0208\text{ lbm/ft}^3)(806.7\text{ ft/s})^2}{2}\left(\frac{1\text{ lbf}}{32.2\text{ lbm}\cdot\text{ft/s}^2}\right) = 5675\text{ lbf}$$

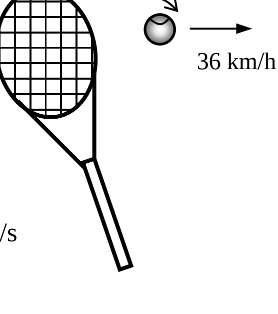
Noting that power is force times velocity (distance per unit time), the power required to overcome this drag is equal to the thrust times the cruising velocity,

$$\text{Power} = \text{Thrust}\times\text{Velocity} = F_D V = (5675\text{ lbf})(806.7\text{ ft/s})\left(\frac{1\text{ kW}}{737.56\text{ lbf}\cdot\text{ft/s}}\right) = \mathbf{6200\text{ kW}}$$

***Discussion*** Note that the engines must supply 6200 kW of power to overcome the drag during cruising. This is the power required to overcome the drag that acts on the wings only, and does not include the drag that acts on the remaining parts of the aircraft (the fuselage, the tail, etc).

## 11-102

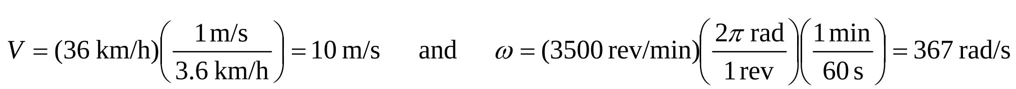
**Solution** A smooth ball is moving at a specified velocity. The increase in the drag coefficient when the ball spins is to be determined.




***Assumptions*** **1** The outer surface of the ball is smooth. **2** The air is calm (no winds or drafts).

***Properties*** The density and kinematic viscosity of air at 1 atm and 25°C are $\rho$ = 1.184 kg/m$^3$ and $\nu$ = 1.562×10$^{-5}$ m$^2$/s.

***Analysis*** Noting that $D$ = 0.08 m and 1 m/s = 3.6 km/h, the regular and angular velocities of the ball are



$$V = (36\text{ km/h})\left(\frac{1\text{ m/s}}{3.6\text{ km/h}}\right) = 10\text{ m/s} \quad \text{and} \quad \omega = (3500\text{ rev/min})\left(\frac{2\pi\text{ rad}}{1\text{ rev}}\right)\left(\frac{1\text{ min}}{60\text{ s}}\right) = 367\text{ rad/s}$$

From these values, we calculate the nondimensional rate of rotation and the Reynolds number:

$$\frac{\omega D}{2V} = \frac{(367\text{ rad/s})(0.08\text{ m})}{2(10\text{ m/s})} = 1.468\text{ rad} \quad \text{and} \quad \text{Re} = \frac{VD}{\nu} = \frac{(10\text{ m/s})(0.08\text{ m})}{1.562\times10^{-5}\text{ m}^2/\text{s}} = 5.122\times10^4$$

Then the drag coefficients for the ball with and without spin are determined from Figs. 11-36 and 11-53 to be:

*Without spin*: $C_D$ = 0.49 (Fig. 11-36, smooth ball)
*With spin*: $C_D$ = 0.58 (Fig. 11-53)

Then the increase in the drag coefficient due to spinning becomes

$$\text{Increase in } C_D = \frac{C_{D,\text{spin}} - C_{D,\text{ no spin}}}{C_{D,\text{ no spin}}} = \frac{0.58 - 0.49}{0.49} = 0.184$$

Therefore, **the drag coefficient in this case increases by about 18.4% because of spinning**.

***Discussion*** Note that the Reynolds number for this problem is 5.122×10$^4$ which is close enough to 6×10$^4$ for which Fig. 11-53 is prepared. Therefore, the result obtained should be fairly accurate.

---

## 11-103

**Solution** The total weight of a paratrooper and its parachute is given. The terminal velocity of the paratrooper in air is to be determined.

***Assumptions*** **1** The air flow over the parachute is turbulent so that the tabulated value of the drag coefficient can be used. **2** The variation of the air properties with altitude is negligible. **3** The buoyancy force applied by air to the person (and the parachute) is negligible because of the small volume occupied and the low air density.

***Properties*** The density of air is given to be 1.20 kg/m$^3$. The drag coefficient of a parachute is $C_D$ = 1.3.

***Analysis*** The terminal velocity of a free falling object is reached when the drag force equals the weight of the solid object, less the buoyancy force applied by the fluid, which is negligible in this case,

$$F_D = W - F_B \quad \text{where} \quad F_D = C_D A \frac{\rho_f V^2}{2}, \quad W = mg = 950\text{ N}, \quad \text{and } F_B \cong 0$$

where $A = \pi D^2/4$ is the frontal area. Substituting and simplifying,

$$C_D A \frac{\rho_f V^2}{2} = W \quad \rightarrow \quad C_D \frac{\pi D^2}{4} \frac{\rho_f V^2}{2} = W$$

Solving for $V$ and substituting,

$$V = \sqrt{\frac{8W}{C_D \pi D^2 \rho_f}} = \sqrt{\frac{8(950\text{ N})}{1.3\pi(8\text{ m})^2(1.20\text{ kg/m}^3)}\left(\frac{1\text{ kg}\cdot\text{m/s}^2}{1\text{ N}}\right)} = \mathbf{4.9\text{ m/s}}$$

Therefore, the velocity of the paratrooper will remain constant when it reaches the terminal velocity of 4.9 m/s = 18 km/h.

***Discussion*** The simple analysis above gives us a rough value for the terminal velocity. A more accurate answer can be obtained by a more detailed (and complex) analysis by considering the variation of air density with altitude, and by considering the uncertainty in the drag coefficient.

---

**11-104**

**Solution** A fairing is installed to the front of a rig to reduce the drag coefficient. The maximum speed of the rig after the fairing is installed is to be determined.

***Assumptions*** **1** The rig moves steadily at a constant velocity on a straight path in calm weather. **2** The bearing friction resistance is constant. **3** The effect of velocity on the drag and rolling resistance coefficients is negligible. **4** The buoyancy of air is negligible. **5** The power produced by the engine is used to overcome rolling resistance, bearing friction, and aerodynamic drag.

***Properties*** The density of air is given to be 1.25 kg/m$^3$. The drag coefficient of the rig is given to be $C_D$ = 0.96, and decreases to $C_D$ = 0.76 when a fairing is installed. The rolling resistance coefficient is $C_{RR}$ = 0.05.

***Analysis*** The bearing friction resistance is given to be $F_{\text{bearing}}$ = 350 N. The rolling resistance is

$$F_{RR} = C_{RR}W = 0.05(17{,}000\text{ kg})(9.81\text{ m/s}^2)\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 8339\text{ N}$$

The maximum drag occurs at maximum velocity, and its value before the fairing is installed is

$$F_{D1} = C_D A\frac{\rho V_1^2}{2} = (0.96)(9.2\text{ m}^2)\frac{(1.25\text{ kg/m}^3)(110/3.6\text{ m/s})^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 5154\text{ N}$$

Power is force times velocity, and thus the power needed to overcome bearing friction, drag, and rolling resistance is the product of the sum of these forces and the velocity of the rig,

$$\begin{aligned}\dot{W}_{\text{total}} &= \dot{W}_{\text{bearing}} + \dot{W}_{\text{drag}} + \dot{W}_{\text{RR}} = (F_{\text{bearing}} + F_D + F_{RR})\mathbf{V}\\ &= (350 + 8339 + 5154)(110/3.6\text{ m/s})\left(\frac{1\text{ kW}}{1000\text{ N}\cdot\text{m/s}}\right)\\ &= 423\text{ kW}\end{aligned}$$

The maximum velocity the rig can attain at the same power of 423 kW after the fairing is installed is determined by setting the sum of the bearing friction, rolling resistance, and the drag force equal to 423 kW,

$$\dot{W}_{\text{total}} = \dot{W}_{\text{bearing}} + \dot{W}_{\text{drag2}} + \dot{W}_{\text{RR}} = (F_{\text{bearing}} + F_{D2} + F_{RR})V_2 = \left(350 + C_{D2}A\frac{\rho V_2^2}{2} + 5154\right)V_2$$

Substituting the known quantities,

$$(423\text{ kW})\left(\frac{1000\text{ N}\cdot\text{m/s}}{1\text{ kW}}\right) = \left(350\text{ N} + (0.76)(9.2\text{ m}^2)\frac{(1.25\text{ kg/m}^3)V_2^2}{2}\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) + 5154\text{ N}\right)V_2$$

or,

$$423{,}000 = 5504V_2 + 4.37V_2^3$$

Solving it with an equation solver gives $V_2$ = **36.9 m/s = 133 km/h**.

***Discussion*** Note that the maximum velocity of the rig increases from 110 km/h to 133 km/h as a result of reducing its drag coefficient from 0.96 to 0.76 while holding the bearing friction and the rolling resistance constant.

**11-105**

**Solution** A spherical object is dropped into a fluid, and its terminal velocity is measured. The viscosity of the fluid is to be determined.

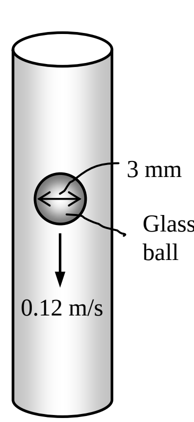
***Assumptions*** **1** The Reynolds number is low (at the order of 1) so that Stokes law is applicable (to be verified). **2** The diameter of the tube that contains the fluid is large enough to simulate free fall in an unbounded fluid body. **3** The tube is long enough to assure that the velocity measured is the terminal velocity.

***Properties*** The density of glass ball is given to be $\rho_s$ = 2500 kg/m$^3$. The density of the fluid is given to be $\rho_f$ = 875 kg/m$^3$.

***Analysis*** The terminal velocity of a free falling object is reached when the drag force equals the weight of the solid object, less the buoyancy force applied by the fluid,

$$F_D = W - F_B \quad \text{where} \quad F_D = 3\pi\mu VD \text{ (Stokes law)}, \quad W = \rho_s g\mathcal{V}, \quad \text{and} \quad F_B = \rho_f g\mathcal{V}$$

Here $\mathcal{V} = \pi D^3/6$ is the volume of the sphere. Substituting and simplifying,




$$3\pi\mu VD = \rho_s g\mathcal{V} - \rho_f g\mathcal{V} \quad \rightarrow \quad 3\pi\mu VD = (\rho_s - \rho_f)g\frac{\pi D^3}{6}$$

Solving for μ and substituting, the dynamic viscosity of the fluid is determined to be

$$\mu = \frac{gD^2(\rho_s - \rho_f)}{18V} = \frac{(9.81\,\text{m/s}^2)(0.003\,\text{m})^2(2500\text{-}875)\,\text{kg/m}^3}{18(0.12\,\text{m/s})} = \mathbf{0.0664\ kg/m \cdot s}$$

The Reynolds number is

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{(875\,\text{kg/m}^3)(0.12\,\text{m/s})(0.003\,\text{m})}{0.0664\,\text{kg}\cdot\text{m/s}} = 4.74$$

which is at the order of 1. Therefore, the creeping flow idealization is valid.

***Discussion*** Flow separation starts at about Re = 10. Therefore, Stokes law can be used for Reynolds numbers up to this value, but this should be done with care.

**11-106**

**Solution** Spherical aluminum balls are dropped into glycerin, and their terminal velocities are measured. The velocities are to be compared to those predicted by Stokes law, and the error involved is to be determined.

***Assumptions*** **1** The Reynolds number is low (at the order of 1) so that Stokes law is applicable (to be verified). **2** The diameter of the tube that contains the fluid is large enough to simulate free fall in an unbounded fluid body. **3** The tube is long enough to assure that the velocity measured is terminal velocity.

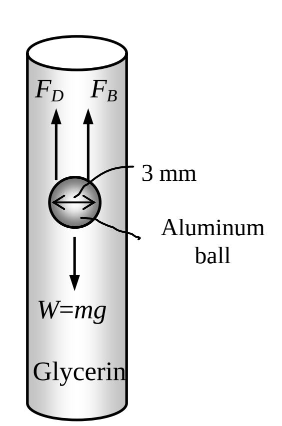
***Properties*** The density of aluminum balls is given to be $\rho_s$ = 2700 kg/m$^3$. The density and viscosity of glycerin are given to be $\rho_f$ = 1274 kg/m$^3$ and $\mu$ = 1 kg/m·s.

***Analysis*** The terminal velocity of a free falling object is reached when the drag force equals the weight of the solid object, less the buoyancy force applied by the fluid,

$$F_D = W - F_B \quad \text{where} \quad F_D = 3\pi\mu VD \text{ (Stokes law)}, \quad W = \rho_s g\mathcal{V}, \quad \text{and } F_B = \rho_f g\mathcal{V}$$

Here $\mathcal{V} = \pi D^3/6$ is the volume of the sphere. Substituting and simplifying,

$$3\pi\mu VD = \rho_s g\mathcal{V} - \rho_f g\mathcal{V} \quad \rightarrow \quad 3\pi\mu VD = (\rho_s - \rho_f)g\frac{\pi D^3}{6}$$




Solving for the terminal velocity $V$ of the ball gives

$$V = \frac{gD^2(\rho_s - \rho_f)}{18\mu}$$

(*a*) $D$ = 2 mm and $V$ = 3.2 mm/s

$$V = \frac{(9.81\text{ m/s}^2)(0.002\text{ m})^2(2700 - 1274)\text{ kg/m}^3}{18(1\text{ kg/m}\cdot\text{s})} = \mathbf{0.00311\text{ m/s}} = \mathbf{3.11\text{ mm/s}}$$

$$\text{Error} = \frac{V_{\text{experimental}} - V_{\text{Stokes}}}{V_{\text{experimental}}} = \frac{3.2 - 3.11}{3.2} = \mathbf{0.029 \quad or\ 2.9\%}$$

(*b*) $D$ = 4 mm and $V$ = 12.8 mm/s

$$V = \frac{(9.81\text{ m/s}^2)(0.004\text{ m})^2(2700 - 1274)\text{ kg/m}^3}{18(1\text{ kg/m}\cdot\text{s})} = \mathbf{0.0124\text{ m/s}} = \mathbf{12.4\text{ mm/s}}$$

$$\text{Error} = \frac{V_{\text{experimental}} - V_{\text{Stokes}}}{V_{\text{experimental}}} = \frac{12.8 - 12.4}{12.8} = \mathbf{0.029 \quad or\ 2.9\%}$$

(*c*) $D$ = 10 mm and $V$ = 60.4 mm/s

$$V = \frac{(9.81\text{ m/s}^2)(0.010\text{ m})^2(2700 - 1274)\text{ kg/m}^3}{18(1\text{ kg/m}\cdot\text{s})} = \mathbf{0.0777\text{ m/s}} = \mathbf{77.7\text{ mm/s}}$$

$$\text{Error} = \frac{V_{\text{experimental}} - V_{\text{Stokes}}}{V_{\text{experimental}}} = \frac{60.4 - 77.7}{60.4} = -\mathbf{0.287 \quad or\ -28.7\%}$$

There is a good agreement for the first two diameters. However the error for third one is large. The Reynolds number for each case is

$$(a)\ \text{Re} = \frac{\rho VD}{\mu} = \frac{(1274\text{ kg/m}^3)(0.0032\text{ m/s})(0.002\text{ m})}{1\text{ kg}\cdot\text{m/s}} = 0.008, \quad (b)\ \text{Re} = 0.065, \quad \text{and} \quad (c)\ \text{Re} = 0.770.$$

We observe that Re << 1 for the first two cases, and thus the creeping flow idealization is applicable. But this is not the case for the third case.

***Discussion*** If we used the general form of the equation (see next problem) we would obtain $V$ = 59.7 mm/s for part (*c*), which is very close to the experimental result (60.4 mm/s).

**11-107**

**Solution** Spherical aluminum balls are dropped into glycerin, and their terminal velocities are measured. The velocities predicted by general form of Stokes law and the error involved are to be determined.

***Assumptions*** **1** The Reynolds number is low (of order 1) so that Stokes law is applicable (to be verified). **2** The diameter of the tube that contains the fluid is large enough to simulate free fall in an unbounded fluid body. **3** The tube is long enough to assure that the velocity measured is terminal velocity.

***Properties*** The density of aluminum balls is given to be $\rho_s$ = 2700 kg/m$^3$. The density and viscosity of glycerin are given to be $\rho_f$ = 1274 kg/m$^3$ and $\mu$ = 1 kg/m·s.

***Analysis*** The terminal velocity of a free falling object is reached when the drag force equals the weight of the solid object, less the buoyancy force applied by the fluid,

$$F_D = W - F_B$$

where $F_D = 3\pi\mu DV + (9\pi/16)\rho_s V^2 D^2$, $W = \rho_s g\mathcal{V}$, and $F_B = \rho_f g\mathcal{V}$

Here $\mathcal{V} = \pi D^3/6$ is the volume of the sphere. Substituting and simplifying,

$$3\pi\mu VD + (9\pi/16)\rho_s V^2 D^2 = (\rho_s - \rho_f)g\frac{\pi D^3}{6}$$

Solving for the terminal velocity $V$ of the ball gives

$$V = \frac{-b + \sqrt{b^2 - 4ac}}{2a} \quad \text{where } a = \frac{9\pi}{16}\rho_s D^2,\ b = 3\pi\mu D, \text{ and } c = -(\rho_s - \rho_f)g\frac{\pi D^3}{6}$$

FD FB
3 mm
Aluminum ball
W=mg
Glycerin

(*a*) $D$ = 2 mm and $V$ = 3.2 mm/s: $a$ = 0.01909, $b$ = 0.01885, $c$ = -0.0000586

$$V = \frac{-0.01885 + \sqrt{(0.01885)^2 - 4\times 0.01909\times(-0.0000586)}}{2\times 0.01909} = \mathbf{0.00310\ m/s} = \mathbf{3.10\ mm/s}$$

$$\text{Error} = \frac{V_{\text{experimental}} - V_{\text{Stokes}}}{V_{\text{experimental}}} = \frac{3.2 - 3.10}{3.2} = \mathbf{0.032 \quad or\ 3.2\%}$$

(*b*) $D$ = 4 mm and $V$ = 12.8 mm/s: $a$ = 0.07634, $b$ = 0.0377, $c$ = -0.0004688

$$V = \frac{-0.0377 + \sqrt{(0.0377)^2 - 4\times 0.07634\times(-0.0004688)}}{2\times 0.07634} = \mathbf{0.0121\ m/s} = \mathbf{12.1\ mm/s}$$

$$\text{Error} = \frac{V_{\text{experimental}} - V_{\text{Stokes}}}{V_{\text{experimental}}} = \frac{12.8 - 12.1}{12.8} = \mathbf{0.052 \quad or\ 5.2\%}$$

(*c*) $D$ = 10 mm and $V$ = 60.4 mm/s: $a$ = 0.4771, $b$ = 0.09425, $c$ = -0.007325

$$V = \frac{-0.09425 + \sqrt{(0.09425)^2 - 4\times 0.3771\times(-0.007325)}}{2\times 0.4771} = \mathbf{0.0597\ m/s} = \mathbf{59.7\ mm/s}$$

$$\text{Error} = \frac{V_{\text{experimental}} - V_{\text{Stokes}}}{V_{\text{experimental}}} = \frac{60.4 - 59.7}{60.4} = \mathbf{0.012 \quad or\ 1.2\%}$$

The Reynolds number for the three cases are

(*a*) $\text{Re} = \dfrac{\rho VD}{\mu} = \dfrac{(1274\ \text{kg/m}^3)(0.0032\ \text{m/s})(0.002\ \text{m})}{1\ \text{kg}\cdot\text{m/s}} = 0.008$, (*b*) Re = 0.065, and (*c*) Re = 0.770.

***Discussion*** There is a good agreement for the third case (case *c*), but the general Stokes law increased the error for the first two cases (cases *a* and *b*) from 2.9% and 2.9% to 3.2% and 5.2%, respectively. Therefore, the basic form of Stokes law should be preferred when the Reynolds number is much lower than 1.

**11-108**

**Solution** A spherical aluminum ball is dropped into oil. A relation is to be obtained for the variation of velocity with time and the terminal velocity of the ball. The variation of velocity with time is to be plotted, and the time it takes to reach 99% of terminal velocity is to be determined.

***Assumptions*** **1** The Reynolds number is low (Re << 1) so that Stokes law is applicable. **2** The diameter of the tube that contains the fluid is large enough to simulate free fall in an unbounded fluid body.

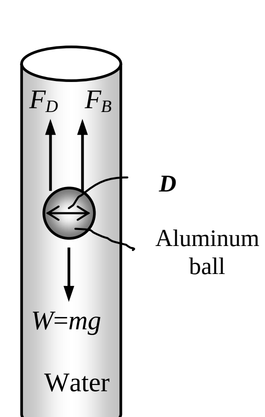
***Properties*** The density of aluminum balls is given to be $\rho_s$ = 2700 kg/m$^3$. The density and viscosity of oil are given to be $\rho_f$ = 876 kg/m$^3$ and $\mu$ = 0.2177 kg/m·s.

***Analysis*** The free body diagram is shown in the figure. The net force acting downward on the ball is the weight of the ball less the weight of the ball and the buoyancy force applied by the fluid,

$$F_{net} = W - F_D - F_B \quad \text{where} \quad F_D = 3\pi\mu DV\,, \quad W = m_s g = \rho_s g\mathcal{V}\,, \quad \text{and} \quad F_B = \rho_f g\mathcal{V}$$

where $F_D$ is the drag force, $F_B$ is the buoyancy force, and $W$ is the weight. Also, $\mathcal{V} = \pi D^3/6$ is the volume, $m_s$ is the mass, $D$ is the diameter, and $V$ the velocity of the ball. Applying Newton's second law in the vertical direction,

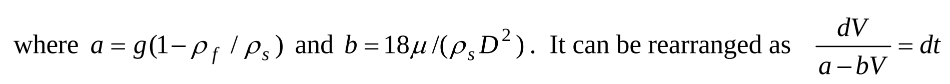
$$F_{net} = ma \quad \rightarrow \quad m_s g - F_D - F_B = m\frac{dV}{dt}$$




Substituting the drag and buoyancy force relations,

$$\rho_s \frac{\pi D^3}{6} g - 3\pi\mu DV - \rho_f g \frac{\pi D^3}{6} = \rho_s \frac{\pi D^3}{6}\frac{dV}{dt}$$

or, $$g\left(1 - \frac{\rho_f}{\rho s}\right) - \frac{18\mu}{\rho_s D^2}V = \frac{dV}{dt} \quad \rightarrow \quad a - bV = \frac{dV}{dt}$$



where $a = g(1 - \rho_f / \rho_s)$ and $b = 18\mu/(\rho_s D^2)$. It can be rearranged as $\dfrac{dV}{a - bV} = dt$

Integrating from $t$ = 0 where $V$ = 0 to $t = t$ where $V = V$ gives

$$\int_0^V \frac{dV}{a - bV} = \int_0^t dt \quad \rightarrow \quad -\frac{\ln(a - bV)}{b}\bigg|_0^V = t\,\big|_0^t \quad \rightarrow \quad \ln\left(\frac{a - bV}{a}\right) = -bt$$

Solving for $V$ gives the desired relation for the variation of velocity of the ball with time,

$$V = \frac{a}{b}\left(1 - e^{-bt}\right) \quad \text{or} \quad V = \frac{(\rho_s - \rho_f)gD^2}{18\mu}\left(1 - e^{-\frac{18\mu}{\rho_s D^2}t}\right) \qquad (1)$$

Note that as $t \rightarrow \infty$, it gives the terminal velocity as $$V_{\text{terminal}} = \frac{a}{b} = \frac{(\rho_s - \rho_f)gD^2}{18\mu} \qquad (2)$$

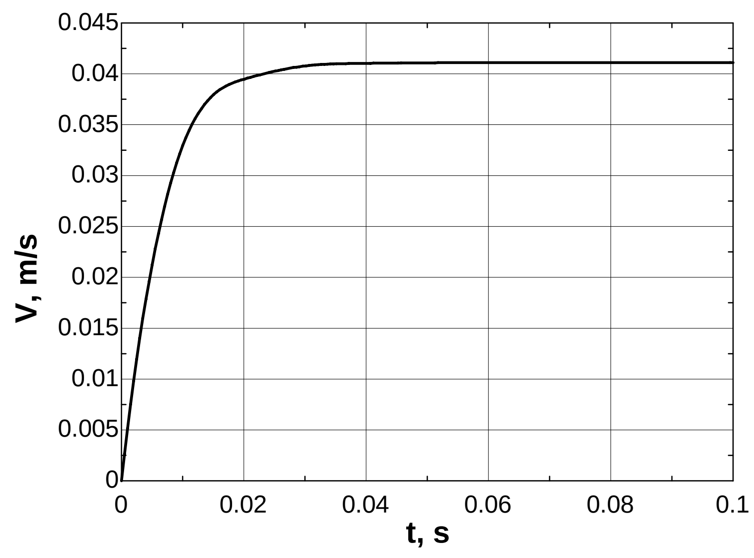
The time it takes to reach 99% of terminal velocity can to be determined by replacing $V$ in Eq. 1 by 0.99$V_{\text{terminal}}$= 0.99$a/b$. This gives $e^{-bt}$ = 0.01 or

$$t_{99\%} = -\frac{\ln(0.01)}{b} = -\frac{\ln(0.01)\rho_s D^2}{18\mu} \qquad (3)$$

*Given values*: $D$ = 0.003 m, $\rho_f$ = 876 kg/m$^3$, $\mu$ = 0.2177 kg/m·s, $g$ = 9.81 m/s$^2$.

*Calculation results*: Re = 0.50, $a$ = 6.627, $b$ = 161.3, $t_{99\%}$ = **0.029 s**, and $V_{\text{terminal}}$= $a/b$ = **0.04 m/s**.

| *t*, s | *V*, m/s |
|---|---|
| 0.00 | 0.000 |
| 0.01 | 0.033 |
| 0.02 | 0.039 |
| 0.03 | 0.041 |
| 0.04 | 0.041 |
| 0.05 | 0.041 |
| 0.06 | 0.041 |
| 0.07 | 0.041 |
| 0.08 | 0.041 |
| 0.09 | 0.041 |
| 0.10 | 0.041 |




***Discussion*** The velocity increases rapidly at first, but quickly levels off by around 0.04 s.



## 11-109



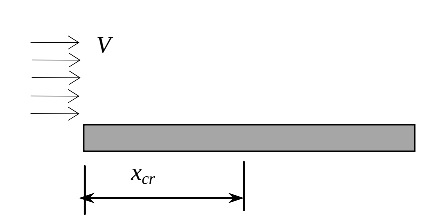
**Solution** Engine oil flows over a long flat plate. The distance from the leading edge $x_{cr}$ where the flow becomes turbulent is to be determined, and thickness of the boundary layer over a distance of $2x_{cr}$ is to be plotted.

***Assumptions*** **1** The flow is steady and incompressible. **2** The critical Reynolds number is $Re_{cr} = 5\times10^5$. **3** The surface of the plate is smooth.

***Properties*** The kinematic viscosity of engine oil at 40°C is $\nu = 2.485\times10^{-4}$ m²/s.

***Analysis*** The thickness of the boundary layer along the flow for laminar and turbulent flows is given by

$$\text{Laminar flow:}\quad \delta_x = \frac{4.91x}{\mathrm{Re}_x^{1/2}}, \qquad \text{Turbulent flow:}\quad \delta_x = \frac{0.38x}{\mathrm{Re}_x^{1/5}}$$




The distance from the leading edge $x_{cr}$ where the flow turns turbulent is determined by setting Reynolds number equal to the critical Reynolds number,

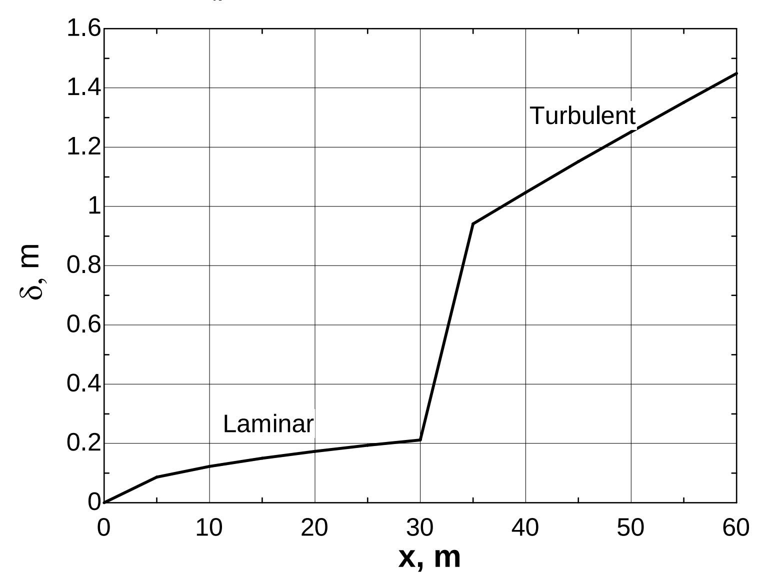
$$\mathrm{Re}_{cr} = \frac{Vx_{cr}}{\nu} \quad\rightarrow\quad x_{cr} = \frac{\mathrm{Re}_{cr}\nu}{V} = \frac{(5\times10^5)(2.485\times10^{-4}\ \mathrm{m^2/s})}{4\ \mathrm{m/s}} = \mathbf{31.1\,m},$$

Therefore, we should consider flow over 2×31.1 = 62.2 m long section of the plate, and use the laminar relation for the first half, and the turbulent relation for the second part to determine the boundary layer thickness. For example, the Reynolds number and the boundary layer thickness at a distance 2 m from the leading edge of the plate are

$$\mathrm{Re}_x = \frac{Vx}{\nu} = \frac{(4\ \mathrm{m/s})(2\ \mathrm{m})}{2.485\times10^{-4}\ \mathrm{m^2/s}} = 32{,}190, \qquad \delta_x = \frac{4.91x}{\mathrm{Re}_x^{1/2}} = \frac{4.91\times(2\ \mathrm{m})}{(32{,}190)^{0.5}} = 0.0547\ \mathrm{m}$$

Calculating the boundary layer thickness and plotting give

| $x$, m | Re | $\delta_{x,\ \text{laminar}}$ | $\delta_{x,\ \text{turbulent}}$ |
|---|---|---|---|
| 0.00 | 0 | 0.000 | - |
| 5.00 | 8.05E+04 | 0.087 | - |
| 10.00 | 1.61E+05 | 0.122 | - |
| 15.00 | 2.41E+05 | 0.150 | - |
| 20.00 | 3.22E+05 | 0.173 | - |
| 25.00 | 4.02E+05 | 0.194 | - |
| 30.00 | 4.83E+05 | 0.212 | - |
| 35.00 | 5.63E+05 | - | 0.941 |
| 40.00 | 6.44E+05 | - | 1.047 |
| 45.00 | 7.24E+05 | - | 1.151 |
| 50.00 | 8.05E+05 | - | 1.252 |
| 55.00 | 8.85E+05 | - | 1.351 |
| 60.00 | 9.66E+05 | - | 1.449 |




***Discussion*** Notice the sudden, rapid rise in boundary layer thickness when the boundary layer becomes turbulent.

---

## Design and Essay Problems

---

### 11-110 to 11-113

**Solution** Students' essays and designs should be unique and will differ from each other.



# Solutions Manual for
# Fluid Mechanics: Fundamentals and Applications
# by Çengel & Cimbala

---

# CHAPTER 12
# COMPRESSIBLE FLOW

---

## Stagnation Properties

### 12-1C

**Solution** We are to discuss the temperature change from an airplane's nose to far away from the aircraft.

***Analysis*** The temperature of the air **rises as it approaches the nose because of the stagnation process**.

***Discussion*** In the frame of reference moving with the aircraft, the air decelerates from high speed to zero at the nose (stagnation point), and this causes the air temperature to rise.

### 12-2C

**Solution** We are to define and discuss stagnation enthalpy.

***Analysis*** *Stagnation enthalpy* **combines the ordinary enthalpy and the kinetic energy of a fluid**, and offers convenience when analyzing high-speed flows. **It differs from the ordinary enthalpy by the kinetic energy term**.

***Discussion*** Most of the time, we mean *specific enthalpy*, i.e., enthalpy per unit mass, when we use the term *enthalpy*.

### 12-3C

**Solution** We are to define dynamic temperature.

***Analysis*** *Dynamic temperature* is **the temperature rise of a fluid during a stagnation process**.

***Discussion*** When a gas decelerates from high speed to zero speed at a stagnation point, the temperature of the gas rises.

### 12-4C

**Solution** We are to discuss the measurement of flowing air temperature with a probe – is there significant error?

***Analysis*** **No, there is not significant error**, because the velocities encountered in air-conditioning applications are very low, and thus the static and the stagnation temperatures are practically identical.

***Discussion*** If the air stream were supersonic, however, the error would indeed be significant.

### 12-5

**Solution** The state of air and its velocity are specified. The stagnation temperature and stagnation pressure of air are to be determined.

***Assumptions*** **1** The stagnation process is isentropic. **2** Air is an ideal gas.

***Properties*** The properties of air at room temperature are $c_p$ = 1.005 kJ/kg·K and $k$ = 1.4.

***Analysis*** The stagnation temperature of air is determined from

$$T_0 = T + \frac{V^2}{2c_p} = 245.9\text{ K} + \frac{(470\text{ m/s})^2}{2\times 1.005\text{ kJ/kg}\cdot\text{K}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 355.8\text{ K} \cong \mathbf{356\ K}$$

Other stagnation properties at the specified state are determined by considering an isentropic process between the specified state and the stagnation state,

$$P_0 = P\left(\frac{T_0}{T}\right)^{k/(k-1)} = (44\text{ kPa})\left(\frac{355.8\text{ K}}{245.9\text{ K}}\right)^{1.4/(1.4-1)} = 160.3\text{ kPa} \cong \mathbf{160\ kPa}$$

***Discussion*** Note that the stagnation properties can be significantly different than thermodynamic properties.

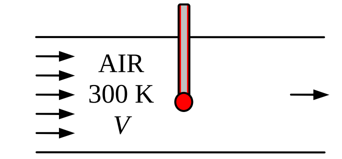
## 12-6

**Solution** Air at 300 K is flowing in a duct. The temperature that a stationary probe inserted into the duct will read is to be determined for different air velocities.

***Assumptions*** The stagnation process is isentropic.

***Properties*** The specific heat of air at room temperature is $c_p$ = 1.005 kJ/kg·K.

***Analysis*** The air which strikes the probe will be brought to a complete stop, and thus it will undergo a stagnation process. The thermometer will sense the temperature of this stagnated air, which is the stagnation temperature, $T_0$. It is determined from $T_0 = T + \frac{V^2}{2c_p}$. The results for each case are calculated below:




(*a*) $$T_0 = 300\text{ K} + \frac{(1\text{ m/s})^2}{2\times 1.005\text{ kJ/kg}\cdot\text{K}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = \mathbf{300.0\text{ K}}$$

(*b*) $$T_0 = 300\text{ K} + \frac{(10\text{ m/s})^2}{2\times 1.005\text{ kJ/kg}\cdot\text{K}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = \mathbf{300.1\text{ K}}$$

(*c*) $$T_0 = 300\text{ K} + \frac{(100\text{ m/s})^2}{2\times 1.005\text{ kJ/kg}\cdot\text{K}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = \mathbf{305.0\text{ K}}$$

(*d*) $$T_0 = 300\text{ K} + \frac{(1000\text{ m/s})^2}{2\times 1.005\text{ kJ/kg}\cdot\text{K}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = \mathbf{797.5\text{ K}}$$

***Discussion*** Note that the stagnation temperature is nearly identical to the thermodynamic temperature at low velocities, but the difference between the two is significant at high velocities.

---

## 12-7

**Solution** The states of different substances and their velocities are specified. The stagnation temperature and stagnation pressures are to be determined.

***Assumptions*** **1** The stagnation process is isentropic. **2** Helium and nitrogen are ideal gases.

***Analysis*** (*a*) Helium can be treated as an ideal gas with $c_p$ = 5.1926 kJ/kg·K and $k$ = 1.667. Then the stagnation temperature and pressure of helium are determined from

$$T_0 = T + \frac{V^2}{2c_p} = 50°\text{C} + \frac{(240\text{ m/s})^2}{2\times 5.1926\text{ kJ/kg}\cdot°\text{C}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = \mathbf{55.5°C}$$

$$P_0 = P\left(\frac{T_0}{T}\right)^{k/(k-1)} = (0.25\text{ MPa})\left(\frac{328.7\text{ K}}{323.2\text{ K}}\right)^{1.667/(1.667-1)} = \mathbf{0.261\text{ MPa}}$$

(*b*) Nitrogen can be treated as an ideal gas with $c_p$ = 1.039 kJ/kg·K and $k$ =1.400. Then the stagnation temperature and pressure of nitrogen are determined from

$$T_0 = T + \frac{V^2}{2c_p} = 50°\text{C} + \frac{(300\text{ m/s})^2}{2\times 1.039\text{ kJ/kg}\cdot°\text{C}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = \mathbf{93.3°C}$$

$$P_0 = P\left(\frac{T_0}{T}\right)^{k/(k-1)} = (0.15\text{ MPa})\left(\frac{366.5\text{ K}}{323.2\text{ K}}\right)^{1.4/(1.4-1)} = \mathbf{0.233\text{ MPa}}$$

(*c*) Steam can be treated as an ideal gas with $c_p$ = 1.865 kJ/kg·K and $k$ =1.329. Then the stagnation temperature and pressure of steam are determined from

$$T_0 = T + \frac{V^2}{2c_p} = 350°\text{C} + \frac{(480\text{ m/s})^2}{2\times 1.865\text{ kJ/kg}\cdot°\text{C}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = \mathbf{411.8°C} = \mathbf{685\text{ K}}$$

$$P_0 = P\left(\frac{T_0}{T}\right)^{k/(k-1)} = (0.1\text{ MPa})\left(\frac{685\text{ K}}{623.2\text{ K}}\right)^{1.329/(1.329-1)} = \mathbf{0.147\text{ MPa}}$$

***Discussion*** Note that the stagnation properties can be significantly different than thermodynamic properties.

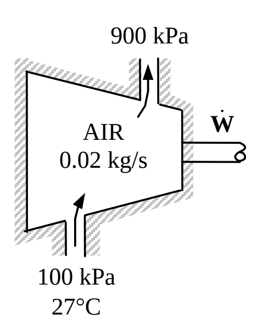
---

**12-8**

**Solution** The inlet stagnation temperature and pressure and the exit stagnation pressure of air flowing through a compressor are specified. The power input to the compressor is to be determined.

***Assumptions*** **1** The compressor is isentropic. **2** Air is an ideal gas.

***Properties*** The properties of air at room temperature are $c_p$ = 1.005 kJ/kg·K and $k$ = 1.4.




***Analysis*** The exit stagnation temperature of air $T_{02}$ is determined from

$$T_{02} = T_{01}\left(\frac{P_{02}}{P_{01}}\right)^{(k-1)/k} = (300.2\text{ K})\left(\frac{900}{100}\right)^{(1.4-1)/1.4} = 562.4\text{ K}$$

From the energy balance on the compressor,

$$\dot{W}_{\text{in}} = \dot{m}(h_{20} - h_{01})$$

or,

$$\dot{W}_{\text{in}} = \dot{m}c_p(T_{02} - T_{01}) = (0.02\text{ kg/s})(1.005\text{ kJ/kg}\cdot\text{K})(562.4 - 300.2)\text{K} = \mathbf{5.27\text{ kW}}$$

***Discussion*** Note that the stagnation properties can be used conveniently in the energy equation.

---

**12-9E**

**Solution** Steam flows through a device. The stagnation temperature and pressure of steam and its velocity are specified. The static pressure and temperature of the steam are to be determined.

***Assumptions*** **1** The stagnation process is isentropic. **2** Steam is an ideal gas.

***Properties*** Steam can be treated as an ideal gas with $c_p$ = 0.4455 Btu/lbm·R and $k$ =1.329.

***Analysis*** The static temperature and pressure of steam are determined from

$$T = T_0 - \frac{V^2}{2c_p} = 700°\text{F} - \frac{(900\text{ ft/s})^2}{2\times 0.4455\text{ Btu/lbm}\cdot°\text{F}}\left(\frac{1\text{ Btu/lbm}}{25{,}037\text{ ft}^2/\text{s}^2}\right) = \mathbf{663.7°F}$$

$$P = P_0\left(\frac{T}{T_0}\right)^{k/(k-1)} = (120\text{ psia})\left(\frac{1123.7\text{ R}}{1160\text{ R}}\right)^{1.329/(1.329-1)} = \mathbf{105.5\text{ psia}}$$

***Discussion*** Note that the stagnation properties can be significantly different than thermodynamic properties.

---

**12-10**

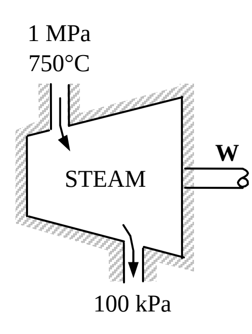
**Solution** The inlet stagnation temperature and pressure and the exit stagnation pressure of products of combustion flowing through a gas turbine are specified. The power output of the turbine is to be determined.

***Assumptions*** **1** The expansion process is isentropic. **2** Products of combustion are ideal gases.

***Properties*** The properties of products of combustion are $c_p$ = 1.157 kJ/kg·K, $R$ = 0.287 kJ/kg·K, and $k$ = 1.33.

***Analysis*** The exit stagnation temperature $T_{02}$ is determined to be

$$T_{02} = T_{01}\left(\frac{P_{02}}{P_{01}}\right)^{(k-1)/k} = (1023.2\text{ K})\left(\frac{0.1}{1}\right)^{(1.33-1)/1.33} = 577.9\text{ K}$$




Also,

$$c_p = kc_v = k\left(c_p - R\right) \longrightarrow c_p = \frac{kR}{k-1}$$

$$= \frac{1.33\left(0.287\text{ kJ/kg}\cdot\text{K}\right)}{1.33-1}$$

$$= 1.157\text{ kJ/kg}\cdot\text{K}$$

From the energy balance on the turbine,

$$-w_{\text{out}} = (h_{20} - h_{01})$$

or, $w_{\text{out}} = c_p\left(T_{01} - T_{02}\right) = \left(1.157\text{ kJ/kg}\cdot\text{K}\right)(1023.2 - 577.9)\text{ K} = 515.2\text{ kJ/kg} \cong \mathbf{515\ kJ/kg}$

***Discussion*** Note that the stagnation properties can be used conveniently in the energy equation.

---

**12-11**

**Solution** Air flows through a device. The stagnation temperature and pressure of air and its velocity are specified. The static pressure and temperature of air are to be determined.

***Assumptions*** **1** The stagnation process is isentropic. **2** Air is an ideal gas.

***Properties*** The properties of air at an anticipated average temperature of 600 K are $c_p$ = 1.051 kJ/kg·K and $k$ = 1.376.

***Analysis*** The static temperature and pressure of air are determined from

$$T = T_0 - \frac{V^2}{2c_p} = 673.2 - \frac{(570\text{ m/s})^2}{2\times 1.051\text{ kJ/kg}\cdot\text{K}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = \mathbf{518.6\ K}$$

and

$$P_2 = P_{02}\left(\frac{T_2}{T_{02}}\right)^{k/(k-1)} = (0.6\text{ MPa})\left(\frac{518.6\text{ K}}{673.2\text{ K}}\right)^{1.376/(1.376-1)} = \mathbf{0.23\ MPa}$$

***Discussion*** Note that the stagnation properties can be significantly different than thermodynamic properties.

---

## Speed of sound and Mach Number

**12-12C**

**Solution** We are to define and discuss sound and how it is generated and how it travels.

*Analysis* *Sound* is an **infinitesimally small pressure wave**. It is **generated by a small disturbance in a medium**. It **travels by wave propagation**. **Sound waves cannot travel in a vacuum**.

*Discussion* Electromagnetic waves, like light and radio waves, can travel in a vacuum, but sound cannot.

---

**12-13C**

**Solution** We are to state whether the propagation of sound waves is an isentropic process.

*Analysis* **Yes, the propagation of sound waves is nearly isentropic**. Because the amplitude of an ordinary sound wave is very small, and it does not cause any significant change in temperature and pressure.

*Discussion* No process is truly isentropic, but the increase of entropy due to sound propagation is negligibly small.

---

**12-14C**

**Solution** We are to discuss sonic velocity – specifically, whether it is constant or it changes.

*Analysis* The *sonic speed* in a medium **depends on the properties of the medium, and it changes as the properties of the medium change**.

*Discussion* The most common example is the change in speed of sound due to temperature change.

---

**12-15C**

**Solution** We are to discuss whether sound travels faster in warm or cool air.

*Analysis* Sound travels faster in **warm (higher temperature) air** since $c = \sqrt{kRT}$ .

*Discussion* On the microscopic scale, we can imagine the air molecules moving around at higher speed in warmer air, leading to higher propagation of disturbances.

---

**12-16C**

**Solution** We are to compare the speed of sound in air, helium, and argon.

*Analysis* **Sound travels fastest in helium**, since $c = \sqrt{kRT}$ and helium has the highest *kR* value. It is about 0.40 for air, 0.35 for argon, and 3.46 for helium.

*Discussion* We are assuming, of course, that these gases behave as ideal gases – a good approximation at room temperature.

---

**12-17C**

**Solution** We are to compare the speed of sound in air at two different pressures, but the same temperature.

***Analysis*** Air at specified conditions will behave like an ideal gas, and *the speed of sound in an ideal gas depends on temperature only*. Therefore, **the speed of sound is the same in both mediums**.

***Discussion*** If the temperature were different, however, the speed of sound would be different.

---

**12-18C**

**Solution** We are to examine whether the Mach number remains constant in constant-velocity flow.

***Analysis*** In general, **no**, because the Mach number also depends on the speed of sound in gas, which depends on the temperature of the gas. **The Mach number remains constant only if the temperature *and* the velocity are constant**.

***Discussion*** It turns out that the speed of sound is not a strong function of pressure. In fact, it is not a function of pressure at all for an ideal gas.

---

**12-19**

**Solution** The Mach number of an aircraft and the speed of sound in air are to be determined at two specified temperatures.

***Assumptions*** Air is an ideal gas with constant specific heats at room temperature.

***Properties*** The gas constant of air is $R$ = 0.287 kJ/kg·K. Its specific heat ratio at room temperature is $k$ = 1.4.

***Analysis*** From the definitions of the speed of sound and the Mach number,

$$c = \sqrt{kRT} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(300\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = \mathbf{347\ m/s}$$

and $$\text{Ma} = \frac{V}{c} = \frac{240\text{ m/s}}{347\text{ m/s}} = \mathbf{0.692}$$

($b$) At 1000 K,

$$c = \sqrt{kRT} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(1000\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = \mathbf{634\ m/s}$$

and $$\text{Ma} = \frac{V}{c} = \frac{240\text{ m/s}}{634\text{ m/s}} = \mathbf{0.379}$$

***Discussion*** Note that a constant Mach number does not necessarily indicate constant speed. The Mach number of a rocket, for example, will be increasing even when it ascends at constant speed. Also, the specific heat ratio $k$ changes with temperature, and the accuracy of the result at 1000 K can be improved by using the $k$ value at that temperature (it would give $k$ = 1.386, $c$ = 619 m/s, and Ma = 0.388).

---

**12-20**

**Solution** Carbon dioxide flows through a nozzle. The inlet temperature and velocity and the exit temperature of $CO_2$ are specified. The Mach number is to be determined at the inlet and exit of the nozzle.

***Assumptions*** **1** $CO_2$ is an ideal gas with constant specific heats at room temperature. **2** This is a steady-flow process.

***Properties*** The gas constant of carbon dioxide is $R$ = 0.1889 kJ/kg·K. Its constant pressure specific heat and specific heat ratio at room temperature are $c_p$ = 0.8439 kJ/kg·K and $k$ = 1.288.

***Analysis*** (*a*) At the inlet

$$c_1 = \sqrt{k_1 R T_1} = \sqrt{(1.288)(0.1889 \text{ kJ/kg}\cdot\text{K})(1200 \text{ K})\left(\frac{1000 \text{ m}^2/\text{s}^2}{1 \text{ kJ/kg}}\right)} = 540.3 \text{ m/s}$$

1200 K
50 m/s
Carbon dioxide
400 K

Thus,

$$\text{Ma}_1 = \frac{V_1}{c_1} = \frac{50 \text{ m/s}}{540.3 \text{ m/s}} = \mathbf{0.0925}$$

(*b*) At the exit,

$$c_2 = \sqrt{k_2 R T_2} = \sqrt{(1.288)(0.1889 \text{ kJ/kg}\cdot\text{K})(400 \text{ K})\left(\frac{1000 \text{ m}^2/\text{s}^2}{1 \text{ kJ/kg}}\right)} = 312.0 \text{ m/s}$$

The nozzle exit velocity is determined from the steady-flow energy balance relation,

$$0 = h_2 - h_1 + \frac{V_2^2 - V_1^2}{2} \quad \rightarrow \quad 0 = c_p(T_2 - T_1) + \frac{V_2^2 - V_1^2}{2}$$

$$0 = (0.8439 \text{ kJ/kg}\cdot\text{K})(400 - 1200 \text{ K}) + \frac{V_2^2 - (50 \text{ m/s})^2}{2}\left(\frac{1 \text{ kJ/kg}}{1000 \text{ m}^2/\text{s}^2}\right) \longrightarrow V_2 = 1163 \text{ m/s}$$

Thus,

$$\text{Ma}_2 = \frac{V_2}{c_2} = \frac{1163 \text{ m/s}}{312 \text{ m/s}} = \mathbf{3.73}$$

***Discussion*** The specific heats and their ratio $k$ change with temperature, and the accuracy of the results can be improved by accounting for this variation. Using EES (or another property database):

At 1200 K: $c_p$ = 1.278 kJ/kg·K, $k$ = 1.173 → $c_1$ = 516 m/s, $V_1$ = 50 m/s, $\text{Ma}_1$ = 0.0969

At 400 K: $c_p$ = 0.9383 kJ/kg·K, $k$ = 1.252 → $c_2$ = 308 m/s, $V_2$ = 1356 m/s, $\text{Ma}_2$ = 4.41

Therefore, the constant specific heat assumption results in an error of **4.5%** at the inlet and **15.5%** at the exit in the Mach number, which are significant.

**12-21**

**Solution** Nitrogen flows through a heat exchanger. The inlet temperature, pressure, and velocity and the exit pressure and velocity are specified. The Mach number is to be determined at the inlet and exit of the heat exchanger.

***Assumptions*** **1** $N_2$ is an ideal gas. **2** This is a steady-flow process. **3** The potential energy change is negligible.

***Properties*** The gas constant of $N_2$ is $R$ = 0.2968 kJ/kg·K. Its constant pressure specific heat and specific heat ratio at room temperature are $c_p$ = 1.040 kJ/kg·K and $k$ = 1.4.

***Analysis***
$$c_1 = \sqrt{k_1 R T_1} = \sqrt{(1.400)(0.2968\ \text{kJ/kg}\cdot\text{K})(283\ \text{K})\left(\frac{1000\ \text{m}^2/\text{s}^2}{1\ \text{kJ/kg}}\right)} = 342.9\ \text{m/s}$$

Thus,

$$\text{Ma}_1 = \frac{V_1}{c_1} = \frac{100\ \text{m/s}}{342.9\ \text{m/s}} = \mathbf{0.292}$$

120kJ/kg

150 kPa
10°C
100 m/s

Nitrogen

100 kPa
200 m/s

From the energy balance on the heat exchanger,

$$q_{\text{in}} = c_p(T_2 - T_1) + \frac{V_2^2 - V_1^2}{2}$$

$$120\ \text{kJ/kg} = (1.040\ \text{kJ/kg.°C})(T_2 - 10\text{°C}) + \frac{(200\ \text{m/s})^2 - (100\ \text{m/s})^2}{2}\left(\frac{1\ \text{kJ/kg}}{1000\ \text{m}^2/\text{s}^2}\right)$$

It yields

$T_2$ = 111°C = 384 K

$$c_2 = \sqrt{k_2 R T_2} = \sqrt{(1.4)(0.2968\ \text{kJ/kg}\cdot\text{K})(384\ \text{K})\left(\frac{1000\ \text{m}^2/\text{s}^2}{1\ \text{kJ/kg}}\right)} = 399\ \text{m/s}$$

Thus,

$$\text{Ma}_2 = \frac{V_2}{c_2} = \frac{200\ \text{m/s}}{399\ \text{m/s}} = \mathbf{0.501}$$

***Discussion*** The specific heats and their ratio $k$ change with temperature, and the accuracy of the results can be improved by accounting for this variation. Using EES (or another property database):

At 10°C : $c_p$ = 1.038 kJ/kg·K, $k$ = 1.400 → $c_1$ = 343 m/s, $V_1$ = 100 m/s, $\text{Ma}_1$ = 0.292

At 111°C $c_p$ = 1.041 kJ/kg·K, $k$ = 1.399 → $c_2$ = 399 m/s, $V_2$ = 200 m/s, $\text{Ma}_2$ = 0.501

Therefore, the constant specific heat assumption results in an error of **4.5%** at the inlet and **15.5%** at the exit in the Mach number, which are almost identical to the values obtained assuming constant specific heats.

---

**12-22**

**Solution** The speed of sound in refrigerant-134a at a specified state is to be determined.

***Assumptions*** R-134a is an ideal gas with constant specific heats at room temperature.

***Properties*** The gas constant of R-134a is $R$ = 0.08149 kJ/kg·K. Its specific heat ratio at room temperature is $k$ = 1.108.

***Analysis*** From the ideal-gas speed of sound relation,

$$c = \sqrt{kRT} = \sqrt{(1.108)(0.08149\ \text{kJ/kg}\cdot\text{K})(60+273\ \text{K})\left(\frac{1000\ \text{m}^2/\text{s}^2}{1\ \text{kJ/kg}}\right)} = \mathbf{173\ m/s}$$

***Discussion*** Note that the speed of sound is independent of pressure for ideal gases.

---

**12-23**

**Solution** The Mach number of a passenger plane for specified limiting operating conditions is to be determined.

***Assumptions*** Air is an ideal gas with constant specific heats at room temperature.

***Properties*** The gas constant of air is $R$ = 0.287 kJ/kg·K. Its specific heat ratio at room temperature is $k$ = 1.4.

***Analysis*** From the speed of sound relation

$$c = \sqrt{kRT} = \sqrt{(1.4)(0.287 \text{ kJ/kg}\cdot\text{K})(-60+273 \text{ K})\left(\frac{1000 \text{ m}^2/\text{s}^2}{1 \text{ kJ/kg}}\right)} = 293 \text{ m/s}$$

Thus, the Mach number corresponding to the maximum cruising speed of the plane is

$$\text{Ma} = \frac{V_{\max}}{c} = \frac{(945/3.6) \text{ m/s}}{293 \text{ m/s}} = \mathbf{0.897}$$

***Discussion*** Note that this is a subsonic flight since Ma < 1. Also, using a $k$ value at -60°C would give practically the same result.

---

**12-24E**

**Solution** Steam flows through a device at a specified state and velocity. The Mach number of steam is to be determined assuming ideal gas behavior.

***Assumptions*** Steam is an ideal gas with constant specific heats.

***Properties*** The gas constant of steam is $R$ = 0.1102 Btu/lbm·R. Its specific heat ratio is given to be $k$ = 1.3.

***Analysis*** From the ideal-gas speed of sound relation,

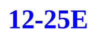
$$c = \sqrt{kRT} = \sqrt{(1.3)(0.1102 \text{ Btu/lbm}\cdot\text{R})(1160 \text{ R})\left(\frac{25{,}037 \text{ ft}^2/\text{s}^2}{1 \text{ Btu/lbm}}\right)} = 2040 \text{ ft/s}$$

Thus,

$$\text{Ma} = \frac{V}{c} = \frac{900 \text{ ft/s}}{2040 \text{ ft/s}} = \mathbf{0.441}$$

***Discussion*** Using property data from steam tables and not assuming ideal gas behavior, it can be shown that the Mach number in steam at the specified state is 0.446, which is sufficiently close to the ideal-gas value of 0.441. Therefore, the ideal gas approximation is a reasonable one in this case.

---

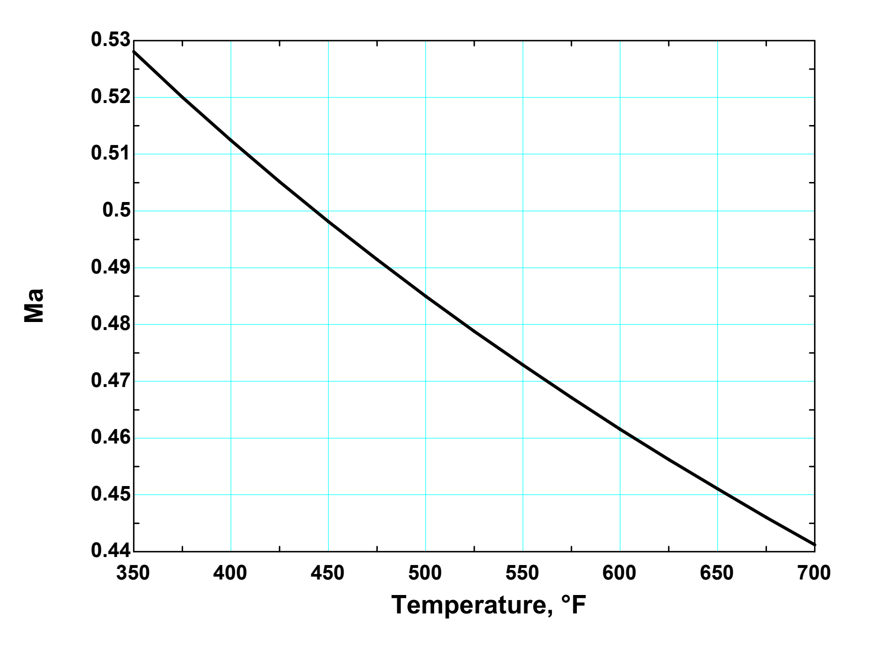

**12-25E** 

**Solution** Problem 12-24e is reconsidered. The variation of Mach number with temperature as the temperature changes between 350° and 700°F is to be investigated, and the results are to be plotted.

***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
T=Temperature+460
R=0.1102
V=900
k=1.3
c=SQRT(k*R*T*25037)
Ma=V/c
```

| Temperature, *T*, °F | Mach number Ma |
|---|---|
| 350 | 0.528 |
| 375 | 0.520 |
| 400 | 0.512 |
| 425 | 0.505 |
| 450 | 0.498 |
| 475 | 0.491 |
| 500 | 0.485 |
| 525 | 0.479 |
| 550 | 0.473 |
| 575 | 0.467 |
| 600 | 0.462 |
| 625 | 0.456 |
| 650 | 0.451 |
| 675 | 0.446 |
| 700 | 0.441 |




***Discussion*** Note that for a specified flow speed, the Mach number decreases with increasing temperature, as expected.

---

**12-26**

**Solution** The expression for the speed of sound for an ideal gas is to be obtained using the isentropic process equation and the definition of the speed of sound.

***Analysis*** The isentropic relation $Pv^k = A$ where $A$ is a constant can also be expressed as

$$P = A\left(\frac{1}{v}\right)^k = A\rho^k$$

Substituting it into the relation for the speed of sound,

$$c^2 = \left(\frac{\partial P}{\partial \rho}\right)_s = \left(\frac{\partial (A\rho)^k}{\partial \rho}\right)_s = kA\rho^{k-1} = k(A\rho^k)/\rho = k(P/\rho) = kRT$$

since for an ideal gas $P = \rho RT$ or $RT = P/\rho$. Therefore, $\boxed{c = \sqrt{kRT}}$, which is the desired relation.

***Discussion*** Notice that pressure has dropped out; the speed of sound in an ideal gas is *not* a function of pressure.

---

**12-27**

**Solution** The inlet state and the exit pressure of air are given for an isentropic expansion process. The ratio of the initial to the final speed of sound is to be determined.

***Assumptions*** Air is an ideal gas with constant specific heats at room temperature.

***Properties*** The properties of air are $R$ = 0.287 kJ/kg·K and $k$ = 1.4. The specific heat ratio k varies with temperature, but in our case this change is very small and can be disregarded.

***Analysis*** The final temperature of air is determined from the isentropic relation of ideal gases,

$$T_2 = T_1\left(\frac{P_2}{P_1}\right)^{(k-1)/k} = (333.2\text{ K})\left(\frac{0.4\text{ MPa}}{1.5\text{ MPa}}\right)^{(1.4-1)/1.4} = 228.4\text{ K}$$

Treating $k$ as a constant, the ratio of the initial to the final speed of sound can be expressed as

$$\text{Ratio} = \frac{c_2}{c_1} = \frac{\sqrt{k_1RT_1}}{\sqrt{k_2RT_2}} = \frac{\sqrt{T_1}}{\sqrt{T_2}} = \frac{\sqrt{333.2}}{\sqrt{228.4}} = \mathbf{1.21}$$

***Discussion*** Note that the speed of sound is proportional to the square root of thermodynamic temperature.

---

**12-28**

**Solution** The inlet state and the exit pressure of helium are given for an isentropic expansion process. The ratio of the initial to the final speed of sound is to be determined.

***Assumptions*** Helium is an ideal gas with constant specific heats at room temperature.

***Properties*** The properties of helium are $R$ = 2.0769 kJ/kg·K and $k$ = 1.667.

***Analysis*** The final temperature of helium is determined from the isentropic relation of ideal gases,

$$T_2 = T_1\left(\frac{P_2}{P_1}\right)^{(k-1)/k} = (333.2\text{ K})\left(\frac{0.4}{1.5}\right)^{(1.667-1)/1.667} = 196.3\text{ K}$$

The ratio of the initial to the final speed of sound can be expressed as

$$\text{Ratio} = \frac{c_2}{c_1} = \frac{\sqrt{k_1RT_1}}{\sqrt{k_2RT_2}} = \frac{\sqrt{T_1}}{\sqrt{T_2}} = \frac{\sqrt{333.2}}{\sqrt{196.3}} = \mathbf{1.30}$$

***Discussion*** Note that the speed of sound is proportional to the square root of thermodynamic temperature.

---

**12-29E**

**Solution** The inlet state and the exit pressure of air are given for an isentropic expansion process. The ratio of the initial to the final speed of sound is to be determined.

***Assumptions*** Air is an ideal gas with constant specific heats at room temperature.

***Properties*** The properties of air are $R$ = 0.06855 Btu/lbm·R and $k$ = 1.4. The specific heat ratio $k$ varies with temperature, but in our case this change is very small and can be disregarded.

***Analysis*** The final temperature of air is determined from the isentropic relation of ideal gases,

$$T_2 = T_1\left(\frac{P_2}{P_1}\right)^{(k-1)/k} = (659.7\text{ R})\left(\frac{60}{170}\right)^{(1.4-1)/1.4} = 489.9\text{ R}$$

Treating k as a constant, the ratio of the initial to the final speed of sound can be expressed as

$$\text{Ratio} = \frac{c_2}{c_1} = \frac{\sqrt{k_1RT_1}}{\sqrt{k_2RT_2}} = \frac{\sqrt{T_1}}{\sqrt{T_2}} = \frac{\sqrt{659.7}}{\sqrt{489.9}} = \mathbf{1.16}$$

***Discussion*** Note that the speed of sound is proportional to the square root of thermodynamic temperature.

---

## One Dimensional Isentropic Flow

**12-30C**

**Solution** We are to discuss what happens to the exit velocity and mass flow rate through a converging nozzle at sonic exit conditions when the nozzle exit area is reduced.

***Analysis*** (*a*) The **exit velocity remains constant at sonic speed**, (*b*) the **mass flow rate through the nozzle decreases because of the reduced flow area**.

***Discussion*** Without a diverging portion of the nozzle, a converging nozzle is limited to sonic velocity at the exit.

**12-31C**

**Solution** We are to discuss what happens to several variables when a supersonic gas enters a converging duct.

***Analysis*** (*a*) The **velocity decreases**. (*b*), (*c*), (*d*) The **temperature, pressure, and density of the fluid increase**.

***Discussion*** The velocity decrease is opposite to what happens in subsonic flow.

**12-32C**

**Solution** We are to discuss what happens to several variables when a supersonic gas enters a diverging duct.

***Analysis*** (*a*) The **velocity increases**. (*b*), (*c*), (*d*) The **temperature, pressure, and density of the fluid decrease**.

***Discussion*** The velocity increase is opposite to what happens in subsonic flow.

**12-33C**

**Solution** We are to discuss what happens to several variables when a subsonic gas enters a converging duct.

***Analysis*** (*a*) The **velocity increases**. (*b*), (*c*), (*d*) The **temperature, pressure, and density of the fluid decrease**.

***Discussion*** The velocity increase is opposite to what happens in supersonic flow.

**12-34C**

**Solution** We are to discuss what happens to several variables when a subsonic gas enters a diverging duct.

***Analysis*** (*a*) The **velocity decreases**. (*b*), (*c*), (*d*) The **temperature, pressure, and density of the fluid increase**.

***Discussion*** The velocity decrease is opposite to what happens in supersonic flow.

**12-35C**

**Solution** We are to discuss the pressure at the throats of two different converging-diverging nozzles.

***Analysis*** The pressure at the two throats are identical.

***Discussion*** Since the gas has the same stagnation conditions, it also has the same sonic conditions at the throat.

**12-36C**

**Solution** We are to determine if it is possible to accelerate a gas to supersonic velocity in a converging nozzle.

***Analysis*** No, it is not possible.

***Discussion*** The only way to do it is to have first a converging nozzle, and then a diverging nozzle.

---

**12-37**

**Solution** Air enters a converging-diverging nozzle at specified conditions. The lowest pressure that can be obtained at the throat of the nozzle is to be determined.

***Assumptions*** **1** Air is an ideal gas with constant specific heats at room temperature. **2** Flow through the nozzle is steady, one-dimensional, and isentropic.

***Properties*** The specific heat ratio of air at room temperature is $k = 1.4$.

***Analysis*** The lowest pressure that can be obtained at the throat is the critical pressure $P^*$, which is determined from

$$P^* = P_0\left(\frac{2}{k+1}\right)^{k/(k-1)} = (1.2\text{ MPa})\left(\frac{2}{1.4+1}\right)^{1.4/(1.4-1)} = \mathbf{0.634\ MPa}$$

***Discussion*** This is the pressure that occurs at the throat when the flow past the throat is supersonic.

---

**12-38**

**Solution** Helium enters a converging-diverging nozzle at specified conditions. The lowest temperature and pressure that can be obtained at the throat of the nozzle are to be determined.

***Assumptions*** **1** Helium is an ideal gas with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic.

***Properties*** The properties of helium are $k = 1.667$ and $c_p = 5.1926$ kJ/kg·K.

***Analysis*** The lowest temperature and pressure that can be obtained at the throat are the critical temperature $T^*$ and critical pressure $P^*$. First we determine the stagnation temperature $T_0$ and stagnation pressure $P_0$,

$$T_0 = T + \frac{V^2}{2c_p} = 800\text{ K} + \frac{(100\text{ m/s})^2}{2\times 5.1926\text{ kJ/kg}\cdot°\text{C}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 801\text{ K}$$

$$P_0 = P\left(\frac{T_0}{T}\right)^{k/(k-1)} = (0.7\text{ MPa})\left(\frac{801\text{ K}}{800\text{ K}}\right)^{1.667/(1.667-1)} = 0.702\text{ MPa}$$




Thus,

$$T^* = T_0\left(\frac{2}{k+1}\right) = (801\text{ K})\left(\frac{2}{1.667+1}\right) = \mathbf{601\ K}$$

and

$$P^* = P_0\left(\frac{2}{k+1}\right)^{k/(k-1)} = (0.702\text{ MPa})\left(\frac{2}{1.667+1}\right)^{1.667/(1.667-1)} = \mathbf{0.342\ MPa}$$

***Discussion*** These are the temperature and pressure that will occur at the throat when the flow past the throat is supersonic.

---

**12-39**

**Solution** The critical temperature, pressure, and density of air and helium are to be determined at specified conditions.

***Assumptions*** Air and Helium are ideal gases with constant specific heats at room temperature.

***Properties*** The properties of air at room temperature are $R$ = 0.287 kJ/kg·K, $k$ = 1.4, and $c_p$ = 1.005 kJ/kg·K. The properties of helium at room temperature are $R$ = 2.0769 kJ/kg·K, $k$ = 1.667, and $c_p$ = 5.1926 kJ/kg·K.

***Analysis*** (*a*) Before we calculate the critical temperature $T^*$, pressure $P^*$, and density $\rho^*$, we need to determine the stagnation temperature $T_0$, pressure $P_0$, and density $\rho_0$.

$$T_0 = 100°\text{C} + \frac{V^2}{2c_p} = 100 + \frac{(250\text{ m/s})^2}{2 \times 1.005\text{ kJ/kg}\cdot°\text{C}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 131.1°\text{C}$$

$$P_0 = P\left(\frac{T_0}{T}\right)^{k/(k-1)} = (200\text{ kPa})\left(\frac{404.3\text{ K}}{373.2\text{ K}}\right)^{1.4/(1.4-1)} = 264.7\text{ kPa}$$

$$\rho_0 = \frac{P_0}{RT_0} = \frac{264.7\text{ kPa}}{(0.287\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(404.3\text{ K})} = 2.281\text{ kg/m}^3$$

Thus,

$$T^* = T_0\left(\frac{2}{k+1}\right) = (404.3\text{ K})\left(\frac{2}{1.4+1}\right) = \mathbf{337\ K}$$

$$P^* = P_0\left(\frac{2}{k+1}\right)^{k/(k-1)} = (264.7\text{ kPa})\left(\frac{2}{1.4+1}\right)^{1.4/(1.4-1)} = \mathbf{140\ kPa}$$

$$\rho^* = \rho_0\left(\frac{2}{k+1}\right)^{1/(k-1)} = (2.281\text{ kg/m}^3)\left(\frac{2}{1.4+1}\right)^{1/(1.4-1)} = \mathbf{1.45\ kg/m^3}$$

(*b*) For helium, $T_0 = T + \frac{V^2}{2c_p} = 40 + \frac{(300\text{ m/s})^2}{2 \times 5.1926\text{ kJ/kg}\cdot°\text{C}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 48.7°\text{C}$

$$P_0 = P\left(\frac{T_0}{T}\right)^{k/(k-1)} = (200\text{ kPa})\left(\frac{321.9\text{ K}}{313.2\text{ K}}\right)^{1.667/(1.667-1)} = 214.2\text{ kPa}$$

$$\rho_0 = \frac{P_0}{RT_0} = \frac{214.2\text{ kPa}}{(2.0769\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(321.9\text{ K})} = 0.320\text{ kg/m}^3$$

Thus,

$$T^* = T_0\left(\frac{2}{k+1}\right) = (321.9\text{ K})\left(\frac{2}{1.667+1}\right) = \mathbf{241\ K}$$

$$P^* = P_0\left(\frac{2}{k+1}\right)^{k/(k-1)} = (200\text{ kPa})\left(\frac{2}{1.667+1}\right)^{1.667/(1.667-1)} = \mathbf{97.4\ kPa}$$

$$\rho^* = \rho_0\left(\frac{2}{k+1}\right)^{1/(k-1)} = (0.320\text{ kg/m}^3)\left(\frac{2}{1.667+1}\right)^{1/(1.667-1)} = \mathbf{0.208\ kg/m^3}$$

***Discussion*** These are the temperature, pressure, and density values that will occur at the throat when the flow past the throat is supersonic.

**12-40**

**Solution** Quiescent carbon dioxide at a given state is accelerated isentropically to a specified Mach number. The temperature and pressure of the carbon dioxide after acceleration are to be determined.

***Assumptions*** Carbon dioxide is an ideal gas with constant specific heats at room temperature.

***Properties*** The specific heat ratio of the carbon dioxide at room temperature is $k$ = 1.288.

***Analysis*** The inlet temperature and pressure in this case is equivalent to the stagnation temperature and pressure since the inlet velocity of the carbon dioxide is said to be negligible. That is, $T_0 = T_i$ = 400 K and $P_0 = P_i$ = 800 kPa. Then,

$$T = T_0\left(\frac{2}{2+(k-1)\mathrm{Ma}^2}\right) = (400\text{ K})\left(\frac{2}{2+(1.288-1)(0.6)^2}\right) = \mathbf{380\ K}$$

and

$$P = P_0\left(\frac{T}{T_0}\right)^{k/(k-1)} = (800\text{ kPa})\left(\frac{380\text{ K}}{400\text{ K}}\right)^{1.288/(1.288-1)} = \mathbf{636\ kPa}$$

***Discussion*** Note that both the pressure and temperature drop as the gas is accelerated as part of the internal energy of the gas is converted to kinetic energy.

---

**12-41**

**Solution** Air flows through a duct. The state of the air and its Mach number are specified. The velocity and the stagnation pressure, temperature, and density of the air are to be determined.

***Assumptions*** Air is an ideal gas with constant specific heats at room temperature.

***Properties*** The properties of air at room temperature are $R$ = 0.287 kPa.m$^3$/kg.K and $k$ = 1.4.

***Analysis*** The speed of sound in air at the specified conditions is

$$c = \sqrt{kRT} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(373.2\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 387.2\text{ m/s}$$

Thus,

$$V = \mathrm{Ma}\times c = (0.8)(387.2\text{ m/s}) = \mathbf{310\ m/s}$$

AIR

Also,

$$\rho = \frac{P}{RT} = \frac{200\text{ kPa}}{(0.287\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(373.2\text{ K})} = 1.867\text{ kg/m}^3$$

Then the stagnation properties are determined from

$$T_0 = T\left(1+\frac{(k-1)\mathrm{Ma}^2}{2}\right) = (373.2\text{ K})\left(1+\frac{(1.4-1)(0.8)^2}{2}\right) = \mathbf{421\ K}$$

$$P_0 = P\left(\frac{T_0}{T}\right)^{k/(k-1)} = (200\text{ kPa})\left(\frac{421.0\text{ K}}{373.2\text{ K}}\right)^{1.4/(1.4-1)} = \mathbf{305\ kPa}$$

$$\rho_0 = \rho\left(\frac{T_0}{T}\right)^{1/(k-1)} = (1.867\text{ kg/m}^3)\left(\frac{421.0\text{ K}}{373.2\text{ K}}\right)^{1/(1.4-1)} = \mathbf{2.52\ kg/m^3}$$

***Discussion*** Note that both the pressure and temperature drop as the gas is accelerated as part of the internal energy of the gas is converted to kinetic energy.

---

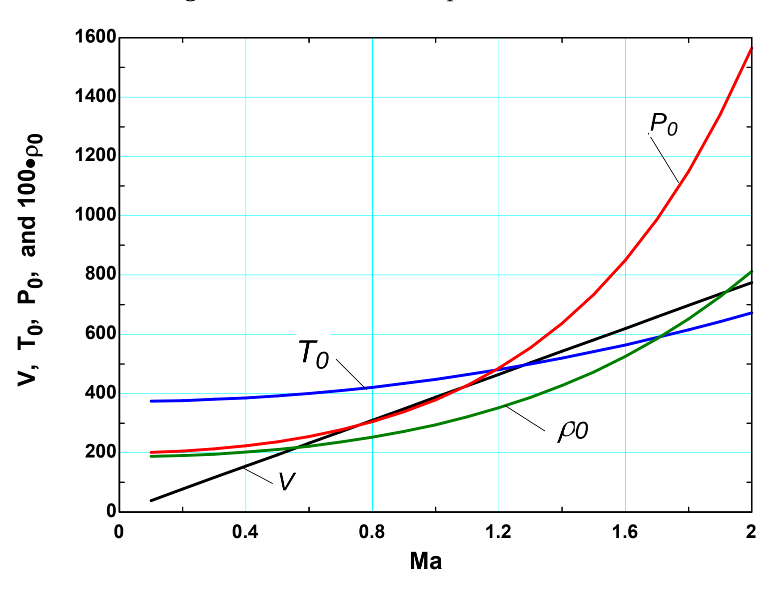

**12-42** 

**Solution** Problem 12-41 is reconsidered. The effect of Mach number on the velocity and stagnation properties as the Ma is varied from 0.1 to 2 are to be investigated, and the results are to be plotted.

***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
P=200
T=100+273.15
R=0.287
k=1.4
c=SQRT(k*R*T*1000)
Ma=V/c
rho=P/(R*T)

"Stagnation properties"
T0=T*(1+(k-1)*Ma^2/2)
P0=P*(T0/T)^(k/(k-1))
rho0=rho*(T0/T)^(1/(k-1))
```




| Mach num. Ma | Velocity, $V$, m/s | Stag. Temp, $T_0$, K | Stag. Press, $P_0$, kPa | Stag. Density, $\rho_0$, kg/m$^3$ |
|---|---|---|---|---|
| 0.1 | 38.7 | 373.9 | 201.4 | 1.877 |
| 0.2 | 77.4 | 376.1 | 205.7 | 1.905 |
| 0.3 | 116.2 | 379.9 | 212.9 | 1.953 |
| 0.4 | 154.9 | 385.1 | 223.3 | 2.021 |
| 0.5 | 193.6 | 391.8 | 237.2 | 2.110 |
| 0.6 | 232.3 | 400.0 | 255.1 | 2.222 |
| 0.7 | 271.0 | 409.7 | 277.4 | 2.359 |
| 0.8 | 309.8 | 420.9 | 304.9 | 2.524 |
| 0.9 | 348.5 | 433.6 | 338.3 | 2.718 |
| 1.0 | 387.2 | 447.8 | 378.6 | 2.946 |
| 1.1 | 425.9 | 463.5 | 427.0 | 3.210 |
| 1.2 | 464.7 | 480.6 | 485.0 | 3.516 |
| 1.3 | 503.4 | 499.3 | 554.1 | 3.867 |
| 1.4 | 542.1 | 519.4 | 636.5 | 4.269 |
| 1.5 | 580.8 | 541.1 | 734.2 | 4.728 |
| 1.6 | 619.5 | 564.2 | 850.1 | 5.250 |
| 1.7 | 658.3 | 588.8 | 987.2 | 5.842 |
| 1.8 | 697.0 | 615.0 | 1149.2 | 6.511 |
| 1.9 | 735.7 | 642.6 | 1340.1 | 7.267 |
| 2.0 | 774.4 | 671.7 | 1564.9 | 8.118 |

***Discussion*** Note that as Mach number increases, so does the flow velocity and stagnation temperature, pressure, and density.

**12-43**

**Solution** Air flows through a duct at a specified state and Mach number. The velocity and the stagnation pressure, temperature, and density of the air are to be determined.

***Assumptions*** Air is an ideal gas with constant specific heats at room temperature.

***Properties*** The properties of air are $R$ = 0.06855 Btu/lbm.R = 0.3704 psia·ft$^3$/lbm.R and $k$ = 1.4.

***Analysis*** The speed of sound in air at the specified conditions is

$$c = \sqrt{kRT} = \sqrt{(1.4)(0.06855\ \text{Btu/1bm}\cdot\text{R})(671.7\ \text{R})\left(\frac{25{,}037\ \text{ft}^2/\text{s}^2}{1\ \text{Btu/1bm}}\right)} = 1270.4\ \text{ft/s}$$

Thus,

$$V = \text{Ma} \times c = (0.8)(1270.4\ \text{ft/s}) = \mathbf{1016\ \ ft/s}$$

Also,

$$\rho = \frac{P}{RT} = \frac{30\ \text{psia}}{(0.3704\ \text{psia}\cdot\text{ft}^3/\text{lbm}\cdot\text{R})(671.7\ \text{R})} = 0.1206\ \text{1bm/ft}^3$$

Then the stagnation properties are determined from

$$T_0 = T\left(1 + \frac{(k-1)\text{Ma}^2}{2}\right) = (671.7\ \text{R})\left(1 + \frac{(1.4\text{-}1)(0.8)^2}{2}\right) = \mathbf{758\ \ R}$$

$$P_0 = P\left(\frac{T_0}{T}\right)^{k/(k-1)} = (30\ \text{psia})\left(\frac{757.7\ \text{R}}{671.7\ \text{R}}\right)^{1.4/(1.4-1)} = \mathbf{45.7\ \ psia}$$

$$\rho_0 = \rho\left(\frac{T_0}{T}\right)^{1/(k-1)} = (0.1206\ \text{1bm/ft}^3)\left(\frac{757.7\ \text{R}}{671.7\ \text{R}}\right)^{1/(1.4-1)} = \mathbf{0.163\ \ 1bm/ft^3}$$

***Discussion*** Note that the temperature, pressure, and density of a gas increases during a stagnation process.

---

**12-44**

**Solution** An aircraft is designed to cruise at a given Mach number, elevation, and the atmospheric temperature. The stagnation temperature on the leading edge of the wing is to be determined.

***Assumptions*** Air is an ideal gas with constant specific heats at room temperature.

***Properties*** The properties of air are $R$ = 0.287 kPa.m$^3$/kg.K, $c_p$ = 1.005 kJ/kg·K, and $k$ = 1.4 .

***Analysis*** The speed of sound in air at the specified conditions is

$$c = \sqrt{kRT} = \sqrt{(1.4)(0.287\ \text{kJ/kg}\cdot\text{K})(236.15\ \text{K})\left(\frac{1000\ \text{m}^2/\text{s}^2}{1\ \text{kJ/kg}}\right)} = 308.0\ \text{m/s}$$

Thus,

$$V = \text{Ma} \times c = (1.4)(308.0\ \text{m/s}) = 431.2\ \text{m/s}$$

Then,

$$T_0 = T + \frac{V^2}{2c_p} = 236.15 + \frac{(431.2\ \text{m/s})^2}{2\times 1.005\ \text{kJ/kg}\cdot\text{K}}\left(\frac{1\ \text{kJ/kg}}{1000\ \text{m}^2/\text{s}^2}\right) = \mathbf{329\ \ K}$$

***Discussion*** Note that the temperature of a gas increases during a stagnation process as the kinetic energy is converted to enthalpy.

---

## Isentropic Flow Through Nozzles

### 12-45C

**Solution** We are to consider subsonic flow through a converging nozzle, and analyze the effect of setting back pressure to critical pressure for a converging nozzle.

***Analysis*** (*a*) The **exit velocity reaches the sonic speed**, (*b*) the **exit pressure equals the critical pressure**, and (*c*) the **mass flow rate reaches the maximum value**.

***Discussion*** In such a case, we say that the flow is *choked*.

### 12-46C

**Solution** We are to consider subsonic flow through a converging nozzle with critical pressure at the exit, and analyze the effect of lowering back pressure below the critical pressure.

***Analysis*** (*a*) **No effect on velocity**. (*b*) **No effect on pressure**. (*c*) **No effect on mass flow rate**.

***Discussion*** In this situation, the flow is already choked initially, so further lowering of the back pressure does not change anything upstream of the nozzle exit plane.

### 12-47C

**Solution** We are to compare the mass flow rates through two identical converging nozzles, but with one having a diverging section.

***Analysis*** **If the back pressure is low enough so that sonic conditions exist at the throats, the mass flow rates in the two nozzles would be identical**. However, **if the flow is not sonic at the throat, the mass flow rate through the nozzle with the diverging section would be greater**, because it acts like a subsonic diffuser.

***Discussion*** Once the flow is choked at the throat, whatever happens downstream is irrelevant to the flow upstream of the throat.

### 12-48C

**Solution** We are to discuss the hypothetical situation of hypersonic flow at the outlet of a converging nozzle.

***Analysis*** Maximum flow rate through a converging nozzle is achieved when Ma = 1 at the exit of a nozzle. For all other Ma values the mass flow rate decreases. Therefore, **the mass flow rate would decrease if hypersonic velocities were achieved at the throat of a converging nozzle**.

***Discussion*** Note that this is not possible unless the flow upstream of the converging nozzle is already hypersonic.

### 12-49C

**Solution** We are to discuss the difference between Ma* and Ma.

***Analysis*** Ma* is the **local velocity non-dimensionalized with respect to the sonic speed at the throat**, whereas Ma is the **local velocity non-dimensionalized with respect to the local sonic speed**.

***Discussion*** The two are identical at the throat when the flow is choked.

**12-50C**

**Solution** We are to discuss what would happen if we add a diverging section to supersonic flow in a duct.

***Analysis*** The fluid would **accelerate even further** instead of decelerating.

***Discussion*** This is the opposite of what would happen in subsonic flow.

---

**12-51C**

**Solution** We are to discuss what would happen if we add a diverging section to supersonic flow in a duct.

***Analysis*** The fluid would **accelerate even further**, as desired.

***Discussion*** This is the opposite of what would happen in subsonic flow.

---

**12-52C**

**Solution** We are to discuss what happens to several variables in the diverging section of a subsonic converging-diverging nozzle.

***Analysis*** (*a*) The **velocity decreases**, (*b*) the **pressure increases**, and (*c*) the **mass flow rate remains the same**.

***Discussion*** Qualitatively, this is the same as what we are used to (in previous chapters) for incompressible flow.

---

**12-53C**

**Solution** We are to analyze if it is possible to accelerate a fluid to supersonic speeds with a velocity that is not sonic at the throat.

***Analysis*** **No, if the flow in the throat is subsonic**. If the velocity at the throat is subsonic, the diverging section would act like a diffuser and decelerate the flow. **Yes, if the flow in the throat is already supersonic**, the diverging section would accelerate the flow to even higher Mach number.

***Discussion*** In duct flow, the latter situation is not possible unless a second converging-diverging portion of the duct is located upstream, and there is sufficient pressure difference to choke the flow in the upstream throat.

---

**12-54**

**Solution** It is to be explained why the maximum flow rate per unit area for a given ideal gas depends only on $P_0/\sqrt{T_0}$. Also for an ideal gas, a relation is to be obtained for the constant *a* in $\dot{m}_{max}/A^* = a\left(P_0/\sqrt{T_0}\right)$.

***Properties*** The properties of the ideal gas considered are $R$ = 0.287 kPa.m$^3$/kg·K and $k$ = 1.4.

***Analysis*** The maximum flow rate is given by

$$\dot{m}_{max} = A^* P_0 \sqrt{k/RT_0}\left(\frac{2}{k+1}\right)^{(k+1)/2(k-1)} \quad \text{or} \quad \dot{m}_{max}/A^* = \left(P_0/\sqrt{T_0}\right)\sqrt{k/R}\left(\frac{2}{k+1}\right)^{(k+1)/2(k-1)}$$

For a given gas, $k$ and $R$ are fixed, and thus the mass flow rate depends on the parameter $P_0/\sqrt{T_0}$. Thus, $\dot{m}_{max}/A^*$ can be expressed as $\dot{m}_{max}/A^* = a\left(P_0/\sqrt{T_0}\right)$ where

$$a = \sqrt{k/R}\left(\frac{2}{k+1}\right)^{(k+1)/2(k-1)} = \sqrt{\frac{1.4}{(0.287\text{ kJ/kg.K})\left(\dfrac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)}}\left(\frac{2}{1.4+1}\right)^{2.4/0.8} = 0.0404\ (\text{m/s})\sqrt{\text{K}}$$

***Discussion*** Note that when sonic conditions exist at a throat of known cross-sectional area, the mass flow rate is fixed by the stagnation conditions.

---

## 12-55

**Solution** For an ideal gas, an expression is to be obtained for the ratio of the speed of sound where Ma = 1 to the speed of sound based on the stagnation temperature, $c^*/c_0$.

***Analysis*** For an ideal gas the speed of sound is expressed as $c = \sqrt{kRT}$ . Thus,

$$\frac{c^*}{c_0} = \frac{\sqrt{kRT^*}}{\sqrt{kRT_0}} = \left(\frac{T^*}{T_0}\right)^{1/2} = \left(\frac{2}{k+1}\right)^{1/2}$$

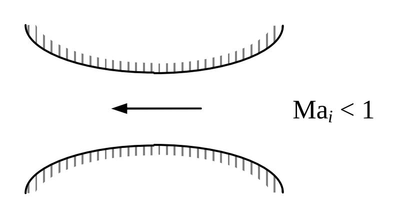
***Discussion*** Note that a speed of sound changes the flow as the temperature changes.

---

## 12-56

**Solution** For subsonic flow at the inlet, the variation of pressure, velocity, and Mach number along the length of the nozzle are to be sketched for an ideal gas under specified conditions.

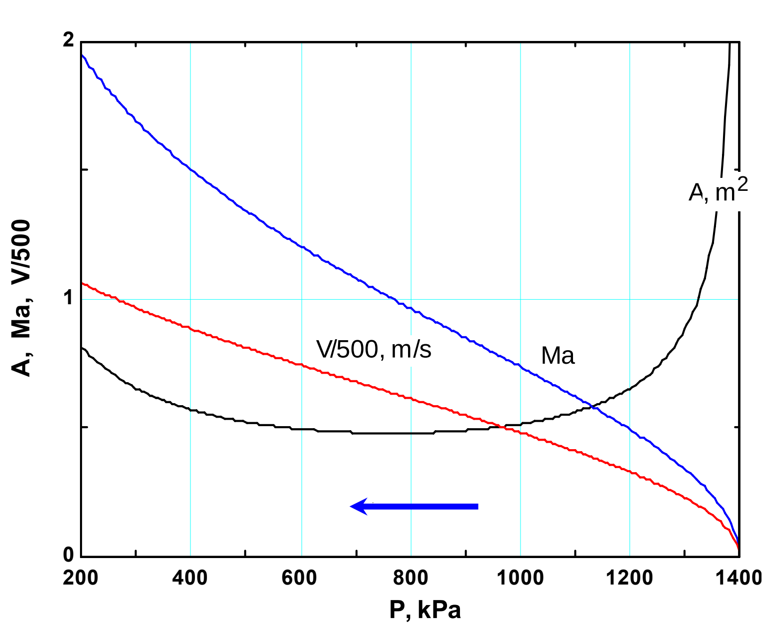
***Assumptions*** **1** The gas is an ideal gas. **2** Flow through the nozzle is steady, one-dimensional, and isentropic. **3** The flow is choked at the throat.




***Analysis*** Using EES and $CO_2$ as the gas, we calculate and plot flow area *A*, velocity *V*, and Mach number Ma as the pressure drops from a stagnation value of 1400 kPa to 200 kPa. Note that the curve for *A* is related to the shape of the nozzle, with horizontal axis serving as the centerline. The EES equation window and the plot are shown below.

```
k=1.289
Cp=0.846 "kJ/kg.K"
R=0.1889 "kJ/kg.K"
P0=1400 "kPa"

T0=473 "K"
m=3 "kg/s"
rho_0=P0/(R*T0)
rho=P/(R*T)
rho_norm=rho/rho_0 "Normalized density"
T=T0*(P/P0)^((k-1)/k)
Tnorm=T/T0  "Normalized temperature"
V=SQRT(2*Cp*(T0-T)*1000)
V_norm=V/500
A=m/(rho*V)*500
C=SQRT(k*R*T*1000)
Ma=V/C
```

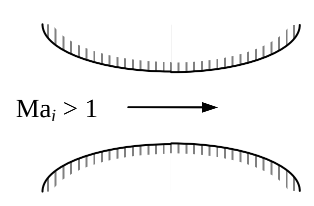



***Discussion*** We are assuming that the back pressure is sufficiently low that the flow is choked at the throat, and the flow downstream of the throat is supersonic without any shock waves. Mach number and velocity continue to rise right through the throat into the diverging portion of the nozzle, since the flow becomes supersonic.

---

**12-57**

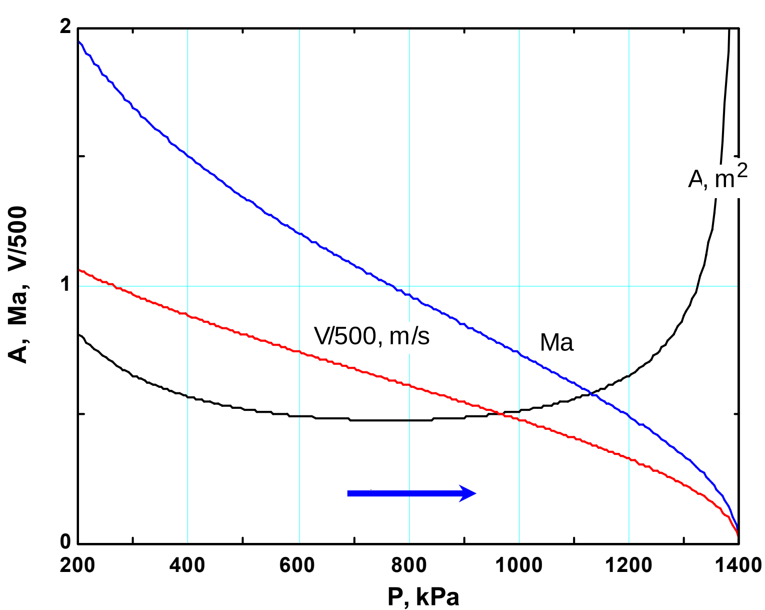
**Solution** We repeat the previous problem, but for supersonic flow at the inlet. The variation of pressure, velocity, and Mach number along the length of the nozzle are to be sketched for an ideal gas under specified conditions.




***Analysis*** Using EES and $CO_2$ as the gas, we calculate and plot flow area *A*, velocity *V*, and Mach number Ma as the pressure rises from 200 kPa at a very high velocity to the stagnation value of 1400 kPa. Note that the curve for *A* is related to the shape of the nozzle, with horizontal axis serving as the centerline.

```
k=1.289
Cp=0.846 "kJ/kg.K"
R=0.1889 "kJ/kg.K"
P0=1400 "kPa"

T0=473 "K"
m=3 "kg/s"
rho_0=P0/(R*T0)
rho=P/(R*T)
rho_norm=rho/rho_0 "Normalized density"
T=T0*(P/P0)^((k-1)/k)
Tnorm=T/T0  "Normalized temperature"
V=SQRT(2*Cp*(T0-T)*1000)
V_norm=V/500
A=m/(rho*V)*500
C=SQRT(k*R*T*1000)
Ma=V/C
```




***Discussion*** Note that this problem is identical to the proceeding one, except the flow direction is reversed. In fact, when plotted like this, the plots are identical.

---

**12-58**

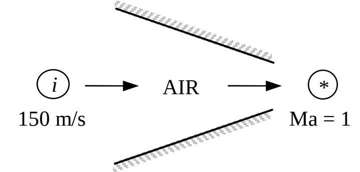
**Solution** Air enters a nozzle at specified temperature, pressure, and velocity. The exit pressure, exit temperature, and exit-to-inlet area ratio are to be determined for a Mach number of Ma = 1 at the exit.

***Assumptions*** **1** Air is an ideal gas with constant specific heats at room temperature. **2** Flow through the nozzle is steady, one-dimensional, and isentropic.

***Properties*** The properties of air are $k$ = 1.4 and $c_p$ = 1.005 kJ/kg·K.

***Analysis*** The properties of the fluid at the location where Ma = 1 are the critical properties, denoted by superscript *. We first determine the stagnation temperature and pressure, which remain constant throughout the nozzle since the flow is isentropic.

$$T_0 = T_i + \frac{V_i^2}{2c_p} = 350\text{ K} + \frac{(150\text{ m/s})^2}{2\times 1.005\text{ kJ/kg}\cdot\text{K}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 361.2\text{ K}$$




and

$$P_0 = P_i\left(\frac{T_0}{T_i}\right)^{k/(k-1)} = (0.2\text{ MPa})\left(\frac{361.2\text{ K}}{350\text{ K}}\right)^{1.4/(1.4-1)} = 0.223\text{ MPa}$$

From Table A-13 (or from Eqs. 12-18 and 12-19) at Ma = 1, we read $T/T_0$ = 0.8333, $P/P_0$ = 0.5283. Thus,

$T = 0.8333T_0$ = 0.8333(361.2 K) = **301 K**

and

$P = 0.5283P_0$ = 0.5283(0.223 MPa) = **0.118 MPa**

Also,

$$c_i = \sqrt{kRT}_i = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(350\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 375\text{ m/s}$$

and

$$\text{Ma}_i = \frac{V_i}{c_i} = \frac{150\text{ m/s}}{375\text{ m/s}} = 0.40$$

From Table A-13 at this Mach number we read $A_i/A^*$ = 1.5901. Thus the ratio of the throat area to the nozzle inlet area is

$$\frac{A^*}{A_i} = \frac{1}{1.5901} = \mathbf{0.629}$$

***Discussion*** We can also solve this problem using the relations for compressible isentropic flow. The results would be identical.

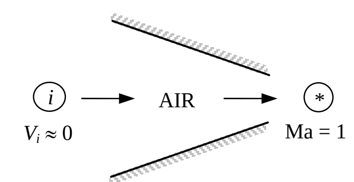
**12-59**

**Solution** Air enters a nozzle at specified temperature and pressure with low velocity. The exit pressure, exit temperature, and exit-to-inlet area ratio are to be determined for a Mach number of Ma = 1 at the exit.

***Assumptions*** **1** Air is an ideal gas. **2** Flow through the nozzle is steady, one-dimensional, and isentropic.

***Properties*** The specific heat ratio of air is $k = 1.4$.

***Analysis*** The properties of the fluid at the location where Ma = 1 are the critical properties, denoted by superscript *. The stagnation temperature and pressure in this case are identical to the inlet temperature and pressure since the inlet velocity is negligible. They remain constant throughout the nozzle since the flow is isentropic.




$T_0 = T_i = 350$ K and $P_0 = P_i = 0.2$ MPa

From Table A-13 (or from Eqs. 12-18 and 12-19) at Ma =1, we read $T/T_0 = 0.8333$, $P/P_0 = 0.5283$. Thus,

$T = 0.8333T_0 = 0.8333(350\text{ K}) = \mathbf{292\text{ K}}$ and $P = 0.5283P_0 = 0.5283(0.2\text{ MPa}) = \mathbf{0.106\text{ MPa}}$

The Mach number at the nozzle inlet is Ma = 0 since $V_i \cong 0$. From Table A-13 at this Mach number we read $A_i/A^* = \infty$. Thus the ratio of the throat area to the nozzle inlet area is $\dfrac{A^*}{A_i} = \dfrac{1}{\infty} = \mathbf{0}$.

***Discussion*** If we solve this problem using the relations for compressible isentropic flow, the results would be identical.

---

**12-60E**

**Solution** Air enters a nozzle at specified temperature, pressure, and velocity. The exit pressure, exit temperature, and exit-to-inlet area ratio are to be determined for a Mach number of Ma = 1 at the exit.

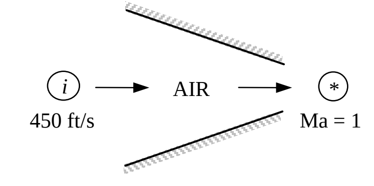
***Assumptions*** **1** Air is an ideal gas with constant specific heats at room temperature. **2** Flow through the nozzle is steady, one-dimensional, and isentropic.

***Properties*** The properties of air are $k = 1.4$ and $c_p = 0.240$ Btu/lbm·R (Table A-2Ea).

***Analysis*** The properties of the fluid at the location where Ma =1 are the critical properties, denoted by superscript *. We first determine the stagnation temperature and pressure, which remain constant throughout the nozzle since the flow is isentropic.

$$T_0 = T + \frac{V_i^2}{2c_p} = 630\text{ R} + \frac{(450\text{ ft/s})^2}{2\times 0.240\text{ Btu/lbm}\cdot\text{R}}\left(\frac{1\text{ Btu/1bm}}{25{,}037\text{ ft}^2/\text{s}^2}\right) = 646.9\text{ R}$$

$$P_0 = P_i\left(\frac{T_0}{T_i}\right)^{k/(k-1)} = (30\text{ psia})\left(\frac{646.9\text{ K}}{630\text{ K}}\right)^{1.4/(1.4-1)} = 32.9\text{ psia}$$




From Table A-13 (or from Eqs. 12-18 and 12-19) at Ma =1, we read $T/T_0 = 0.8333$, $P/P_0 = 0.5283$. Thus,

$T = 0.8333T_0 = 0.8333(646.9\text{ R}) = \mathbf{539\text{ R}}$ and $P = 0.5283P_0 = 0.5283(32.9\text{ psia}) = \mathbf{17.4\text{ psia}}$

Also,

$$c_i = \sqrt{kRT_i} = \sqrt{(1.4)(0.06855\text{ Btu/1bm}\cdot\text{R})(630\text{ R})\left(\frac{25{,}037\text{ ft}^2/\text{s}^2}{1\text{ Btu/1bm}}\right)} = 1230\text{ ft/s} \quad \text{and}$$

$$\text{Ma}_i = \frac{V_i}{c_i} = \frac{450\text{ ft/s}}{1230\text{ ft/s}} = 0.3657$$

From Table A-13 at this Mach number we read $A_i/A^* = 1.7426$. Thus the ratio of the throat area to the nozzle inlet area is

$$\frac{A^*}{A_i} = \frac{1}{1.7426} = \mathbf{0.574}$$

***Discussion*** If we solve this problem using the relations for compressible isentropic flow, the results would be identical.

---

**12-61**

**Solution** Air enters a converging-diverging nozzle at a specified pressure. The back pressure that will result in a specified exit Mach number is to be determined.

***Assumptions*** **1** Air is an ideal gas with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic.

***Properties*** The specific heat ratio of air is $k$ = 1.4.

***Analysis*** The stagnation pressure in this case is identical to the inlet pressure since the inlet velocity is negligible. It remains constant throughout the nozzle since the flow is isentropic,

i AIR e
$V_i \approx 0$ $Ma_e = 1.8$

$$P_0 = P_i = 0.8 \text{ MPa}$$

From Table A-13 at $Ma_e$ =1.8, we read $P_e/P_0$ = 0.1740.

Thus, $P = 0.1740P_0 = 0.1740(0.8 \text{ MPa}) =$ **0.139 MPa**

***Discussion*** If we solve this problem using the relations for compressible isentropic flow, the results would be identical.

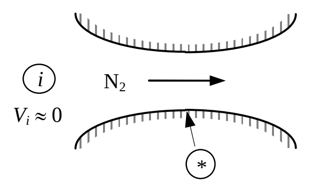
**12-62**

**Solution** Nitrogen enters a converging-diverging nozzle at a given pressure. The critical velocity, pressure, temperature, and density in the nozzle are to be determined.

***Assumptions*** **1** Nitrogen is an ideal gas. **2** Flow through the nozzle is steady, one-dimensional, and isentropic.

***Properties*** The properties of nitrogen are $k$ = 1.4 and $R$ = 0.2968 kJ/kg·K.

***Analysis*** The stagnation pressure in this case are identical to the inlet properties since the inlet velocity is negligible. They remain constant throughout the nozzle,




$$P_0 = P_i = 700 \text{ kPa}$$

$$T_0 = T_i = 400 \text{ K}$$

$$\rho_0 = \frac{P_0}{RT_0} = \frac{700 \text{ kPa}}{(0.2968 \text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(400 \text{ K})} = 5.896 \text{ kg/m}^3$$

Critical properties are those at a location where the Mach number is Ma = 1. From Table A-13 at Ma =1, we read $T/T_0$ =0.8333, $P/P_0$ = 0.5283, and $\rho/\rho_0$ = 0.6339. Then the critical properties become

$T^* = 0.8333T_0 = 0.8333(400 \text{ K}) =$ **333 K**

$P^* = 0.5283P_0 = 0.5283(700 \text{ kPa}) =$ **370 MPa**

$\rho^* = 0.6339\rho_0 = 0.6339(5.896 \text{ kg/m}^3) =$ **3.74 kg/m³**

Also,

$$V^* = c^* = \sqrt{kRT^*} = \sqrt{(1.4)(0.2968 \text{ kJ/kg}\cdot\text{K})(333 \text{ K})\left(\frac{1000 \text{ m}^2/\text{s}^2}{1 \text{ kJ/kg}}\right)} = \mathbf{372 \text{ m/s}}$$

***Discussion*** We can also solve this problem using the relations for compressible isentropic flow. The results would be identical.

**12-63**

**Solution** An ideal gas is flowing through a nozzle. The flow area at a location where Ma = 2.4 is specified. The flow area where Ma = 1.2 is to be determined.

***Assumptions*** Flow through the nozzle is steady, one-dimensional, and isentropic.

***Properties*** The specific heat ratio is given to be $k$ = 1.4.

***Analysis*** The flow is assumed to be isentropic, and thus the stagnation and critical properties remain constant throughout the nozzle. The flow area at a location where $Ma_2$ = 1.2 is determined using $A$ /$A$* data from Table A-13 to be

$$\mathrm{Ma}_1 = 2.4: \frac{A_1}{A^*} = 2.4031 \longrightarrow A^* = \frac{A_1}{2.4031} = \frac{25\ \mathrm{cm}^2}{2.4031} = 10.40\ \mathrm{cm}^2$$

$$\mathrm{Ma}_2 = 1.2: \frac{A_2}{A^*} = 1.0304 \longrightarrow A_2 = (1.0304)A^* = (1.0304)(10.40\ \mathrm{cm}^2) = \mathbf{10.7\ cm^2}$$

***Discussion*** We can also solve this problem using the relations for compressible isentropic flow. The results would be identical.

---

**12-64**

**Solution** An ideal gas is flowing through a nozzle. The flow area at a location where Ma = 2.4 is specified. The flow area where Ma = 1.2 is to be determined.

***Assumptions*** Flow through the nozzle is steady, one-dimensional, and isentropic.

***Analysis*** The flow is assumed to be isentropic, and thus the stagnation and critical properties remain constant throughout the nozzle. The flow area at a location where $Ma_2$ = 1.2 is determined using the $A$ /$A$* relation,

$$\frac{A}{A^*} = \frac{1}{\mathrm{Ma}}\left\{\left(\frac{2}{k+1}\right)\left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)\right\}^{(k+1)/2(k-1)}$$

For $k$ = 1.33 and $Ma_1$ = 2.4:

$$\frac{A_1}{A^*} = \frac{1}{2.4}\left\{\left(\frac{2}{1.33+1}\right)\left(1+\frac{1.33-1}{2}2.4^2\right)\right\}^{2.33/2\times0.33} = 2.570$$

and, $$A^* = \frac{A_1}{2.570} = \frac{25\ \mathrm{cm}^2}{2.570} = 9.729\ \mathrm{cm}^2$$

For $k$ = 1.33 and $Ma_2$ = 1.2:

$$\frac{A_2}{A^*} = \frac{1}{1.2}\left\{\left(\frac{2}{1.33+1}\right)\left(1+\frac{1.33-1}{2}1.2^2\right)\right\}^{2.33/2\times0.33} = 1.0316$$

and $$A_2 = (1.0316)A^* = (1.0316)(9.729\ \mathrm{cm}^2) = \mathbf{10.0\ \ cm^2}$$

***Discussion*** Note that the compressible flow functions in Table A-13 are prepared for $k$ = 1.4, and thus they cannot be used to solve this problem.

---

**12-65** **[*Also solved using EES on enclosed DVD*]**

**Solution** Air enters a converging nozzle at a specified temperature and pressure with low velocity. The exit pressure, the exit velocity, and the mass flow rate versus the back pressure are to be calculated and plotted.

***Assumptions*** **1** Air is an ideal gas with constant specific heats at room temperature. **2** Flow through the nozzle is steady, one-dimensional, and isentropic.

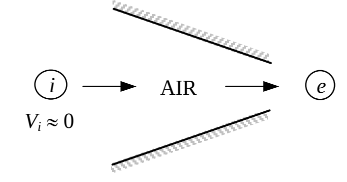
***Properties*** The properties of air are $k = 1.4$, $R = 0.287$ kJ/kg·K, and $c_p = 1.005$ kJ/kg·K.

***Analysis*** The stagnation properties in this case are identical to the inlet properties since the inlet velocity is negligible. They remain constant throughout the nozzle since the flow is isentropic.,

$$P_0 = P_i = 900 \text{ kPa}$$

$$T_0 = T_i = 400 \text{ K}$$

The critical pressure is determined to be



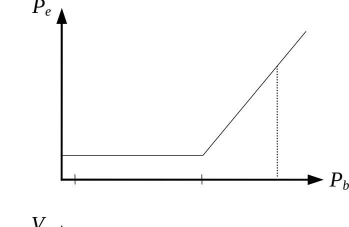

$$P^* = P_0\left(\frac{2}{k+1}\right)^{k/(k-1)} = (900 \text{ kPa})\left(\frac{2}{1.4+1}\right)^{1.4/0.4} = 475.5 \text{ kPa}$$

Then the pressure at the exit plane (throat) will be

$P_e = P_b$ for $P_b \geq 475.5$ kPa

$P_e = P^* = 475.5$ kPa for $P_b < 475.5$ kPa (choked flow)

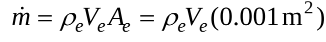
Thus the back pressure will not affect the flow when $100 < P_b < 475.5$ kPa. For a specified exit pressure $P_e$, the temperature, the velocity and the mass flow rate can be determined from




Temperature $$T_e = T_0\left(\frac{P_e}{P_0}\right)^{(k-1)/k} = (400 \text{ K})\left(\frac{P_e}{900}\right)^{0.4/1.4}$$

Velocity $$V = \sqrt{2c_p(T_0 - T_e)} = \sqrt{2(1.005 \text{ kJ/kg}\cdot\text{K})(400 - T_e)\left(\frac{1000 \text{ m}^2/\text{s}^2}{1 \text{ kJ/kg}}\right)}$$

Density $$\rho_e = \frac{P_e}{RT_e} = \frac{P_e}{(0.287 \text{ kPa}\cdot\text{m}^3/kg\cdot K)T_e}$$

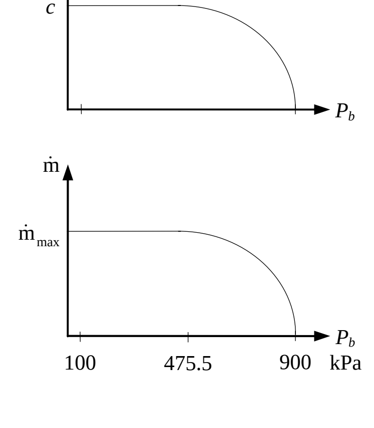


Mass flow rate $$\dot{m} = \rho_e V_e A_e = \rho_e V_e (0.001 \text{ m}^2)$$

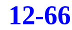


The results of the calculations are tabulated as

| $P_b$, kPa | $P_e$, kPa | $T_e$, K | $V_e$, m/s | $\rho_e$, kg/m³ | $\dot{m}$ kg/s |
|---|---|---|---|---|---|
| 900 | 900 | 400 | 0 | 7.840 | 0 |
| 800 | 800 | 386.8 | 162.9 | 7.206 | 1.174 |
| 700 | 700 | 372.3 | 236.0 | 6.551 | 1.546 |
| 600 | 600 | 356.2 | 296.7 | 5.869 | 1.741 |
| 500 | 500 | 338.2 | 352.4 | 5.151 | 1.815 |
| 475.5 | 475.5 | 333.3 | 366.2 | 4.971 | 1.820 |
| 400 | 475.5 | 333.3 | 366.2 | 4.971 | 1.820 |
| 300 | 475.5 | 333.3 | 366.2 | 4.971 | 1.820 |
| 200 | 475.5 | 333.3 | 366.2 | 4.971 | 1.820 |
| 100 | 475.5 | 333.3 | 366.2 | 4.971 | 1.820 |




***Discussion*** We see from the plots that once the flow is choked at a back pressure of 475.5 kPa, the mass flow rate remains constant regardless of how low the back pressure gets.


**12-66** 

**Solution** We are to reconsider Prob. 12-65. Using EES (or other) software, we are to solve the problem for the inlet conditions of 1 MPa and 1000 K.

***Analysis*** Air at 900 kPa, 400 K enters a converging nozzle with a negligible velocity. The throat area of the nozzle is 10 cm2. Assuming isentropic flow, calculate and plot the exit pressure, the exit velocity, and the mass flow rate versus the back pressure $P_b$ for 0.9>= $P_b$ >=0.1 MPa.

```
Procedure ExitPress(P_back,P_crit : P_exit, Condition$)
If  (P_back>=P_crit)  then
    P_exit:=P_back                    "Unchoked Flow Condition"
    Condition$:='unchoked'
else
    P_exit:=P_crit                    "Choked Flow Condition"
    Condition$:='choked'
Endif
End

"Input data from Diagram Window"
{Gas$='Air'
A_cm2=10                              "Throat area, cm2"
P_inlet = 900"kPa"
T_inlet= 400"K"}
{P_back =475.5  "kPa"}

A_exit = A_cm2*Convert(cm^2,m^2)
C_p=specheat(Gas$,T=T_inlet)
C_p-C_v=R
k=C_p/C_v
M=MOLARMASS(Gas$)                     "Molar mass of  Gas$"
R= 8.314/M                            "Gas constant for Gas$"

"Since the inlet velocity is negligible, the stagnation temperature = T_inlet;
and, since the nozzle is isentropic, the stagnation pressure = P_inlet."

P_o=P_inlet                           "Stagnation pressure"
T_o=T_inlet                           "Stagnation temperature"
P_crit /P_o=(2/(k+1))^(k/(k-1))       "Critical pressure from Eq. 16-22"
Call ExitPress(P_back,P_crit : P_exit, Condition$)

T_exit /T_o=(P_exit/P_o)^((k-1)/k)   "Exit temperature for isentopic flow, K"

V_exit ^2/2=C_p*(T_o-T_exit)*1000    "Exit velocity, m/s"

Rho_exit=P_exit/(R*T_exit)            "Exit density, kg/m3"

m_dot=Rho_exit*V_exit*A_exit          "Nozzle mass flow rate, kg/s"

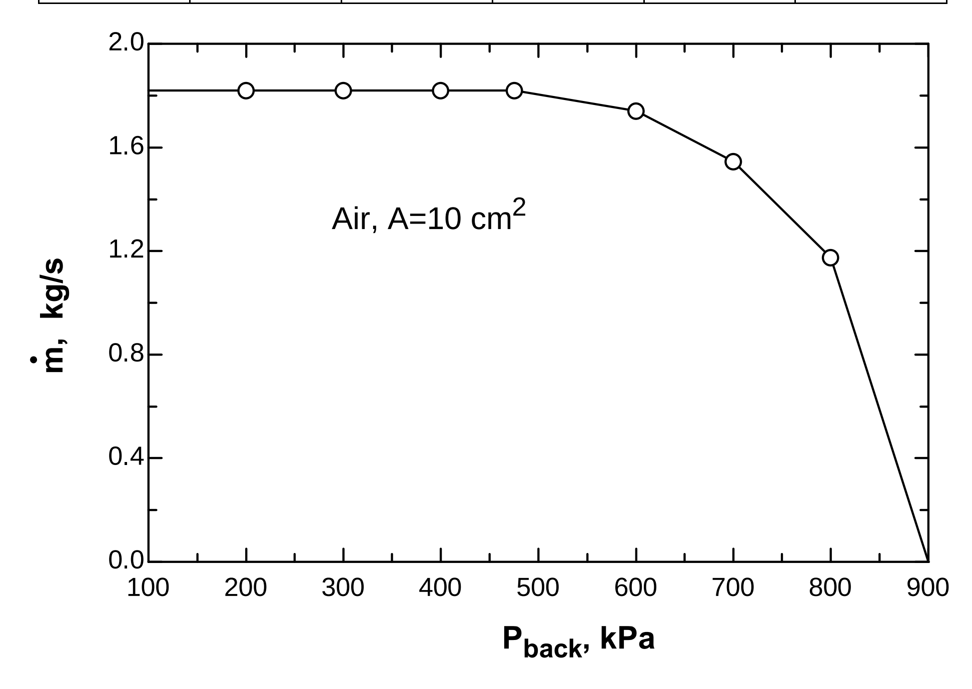
"If you wish to redo the plots, hide the diagram window and remove the { } from
the first 4 variables just under the procedure.  Next set the desired range of
back pressure in the parametric table.  Finally, solve the table (F3).  "
```

The table of results and the corresponding plot are provided below.

**EES SOLUTION**

A_cm2=10 [cm^2]
A_exit=0.001 [m^2]
Condition$='choked'
C_p=1.14 [kJ/kg-K]
C_v=0.8532 [kJ/kg-K]
Gas$='Air'
k=1.336
M=28.97 [kg/kmol]
m_dot=1.258 [kg/s]
P_back=300 [kPA]
P_crit=539.2 [kPA]
P_exit=539.2 [kPA]
P_inlet=1000 [kPA]
P_o=1000 [kPA]
R=0.287 [kJ/kg-K]
Rho_exit=2.195 [m^3/kg]
T_exit=856 [K]
T_inlet=1000 [K]
T_o=1000 [K]
V_exit=573 [m/s]

| m [kg/s] | $P_{exit}$ [kPa] | $T_{exit}$ [K] | $V_{exit}$ [m/s] | $\rho_{exit}$ [kg/m$^3$] | $P_{back}$ [kPa] |
|---|---|---|---|---|---|
| 1.819 | 475.5 | 333.3 | 366.1 | 4.97 | 100 |
| 1.819 | 475.5 | 333.3 | 366.1 | 4.97 | 200 |
| 1.819 | 475.5 | 333.3 | 366.1 | 4.97 | 300 |
| 1.819 | 475.5 | 333.3 | 366.1 | 4.97 | 400 |
| 1.819 | 475.5 | 333.3 | 366 | 4.97 | 475.5 |
| 1.74 | 600 | 356.2 | 296.6 | 5.868 | 600 |
| 1.546 | 700 | 372.3 | 236 | 6.551 | 700 |
| 1.176 | 800 | 386.8 | 163.1 | 7.207 | 800 |
| 0 | 900 | 400 | 0 | 7.839 | 900 |




***Discussion*** We see from the plot that once the flow is choked at a back pressure of 475.5 kPa, the mass flow rate remains constant regardless of how low the back pressure gets.

**12-67E**

**Solution** Air enters a converging-diverging nozzle at a specified temperature and pressure with low velocity. The pressure, temperature, velocity, and mass flow rate are to be calculated in the specified test section.

***Assumptions*** **1** Air is an ideal gas. **2** Flow through the nozzle is steady, one-dimensional, and isentropic.

***Properties*** The properties of air are $k$ = 1.4 and $R$ = 0.06855 Btu/lbm·R = 0.3704 psia·ft$^3$/lbm·R.

***Analysis*** The stagnation properties in this case are identical to the inlet properties since the inlet velocity is negligible. They remain constant throughout the nozzle since the flow is isentropic.

$P_0 = P_i = 150$ psia and $T_0 = T_i = 100°F = 560$ R

Then,

$$T_e = T_0\left(\frac{2}{2+(k-1)\mathrm{Ma}^2}\right) = (560\ \mathrm{R})\left(\frac{2}{2+(1.4\text{-}1)2^2}\right) = \mathbf{311\ R}$$

$$P_e = P_0\left(\frac{T}{T_0}\right)^{k/(k-1)} = (150\ \mathrm{psia})\left(\frac{311}{560}\right)^{1.4/0.4} = \mathbf{19.1\ psia}$$

$$\rho_e = \frac{P_e}{RT_e} = \frac{19.1\ \mathrm{psia}}{(0.3704\ \mathrm{psia.ft^3/1bm\cdot R})(311\ \mathrm{R})} = 0.166\ \mathrm{1bm/ft^3}$$

AIR → ($i$, $V_i \approx 0$; $e$)

The nozzle exit velocity can be determined from $V_e = \mathrm{Ma}_e c_e$ , where $c_e$ is the speed of sound at the exit conditions,

$$V_e = \mathrm{Ma}_e c_e = \mathrm{Ma}_e\sqrt{kRT_e} = (2)\sqrt{(1.4)(0.06855\ \mathrm{Btu/1bm\cdot R})(311\ \mathrm{R})\left(\frac{25{,}037\ \mathrm{ft^2/s^2}}{1\ \mathrm{Btu/1bm}}\right)} = 1729\ \mathrm{ft/s} \cong \mathbf{1730\ ft/s}$$

Finally,

$$\dot{m} = \rho_e A_e V_e = (0.166\ \mathrm{1bm/ft^3})(5\ \mathrm{ft^2})(1729\ \mathrm{ft/s}) = 1435\ \mathrm{lbm/s} \cong \mathbf{1440\ 1bm/s}$$

***Discussion*** Air must be very dry in this application because the exit temperature of air is extremely low, and any moisture in the air will turn to ice particles.

---

## Normal Shocks in Nozzle Flow

**12-68C**

**Solution** We are to discuss if a shock wave can develop in the converging section of a C-V nozzle.

***Analysis*** **No**, because the flow must be supersonic before a shock wave can occur. The flow in the converging section of a nozzle is always subsonic.

***Discussion*** A normal shock (if it is to occur) would occur in the supersonic (diverging) section of the nozzle.

---

**12-69C**

**Solution** We are to discuss the states on the Fanno and Rayleigh lines.

***Analysis*** The *Fanno line* represents the **states that satisfy the conservation of mass and energy equations**. The *Rayleigh line* represents the **states that satisfy the conservation of mass and momentum equations**. The *intersections* points of these lines represent the **states that satisfy the conservation of mass, energy, and momentum equations**.

***Discussion*** *T-s* diagrams are quite helpful in understanding these kinds of flows.

---

**12-70C**

**Solution** We are to determine if Ma downstream of a normal shock can be supersonic.

***Analysis*** **No**, the second law of thermodynamics requires the flow after the shock to be subsonic.

***Discussion*** A normal shock wave always goes from supersonic to subsonic in the flow direction.

---

**12-71C**

**Solution** We are to discuss the effect of a normal shock wave on several properties.

***Analysis*** (*a*) **velocity decreases**, (*b*) **static temperature increases**, (*c*) **stagnation temperature remains the same**, (*d*) **static pressure increases**, and (*e*) **stagnation pressure decreases**.

***Discussion*** In addition, the Mach number goes from supersonic (Ma > 1) to subsonic (Ma < 1).

---

**12-72C**

**Solution** We are to discuss the formation of oblique shocks and how they differ from normal shocks.

***Analysis*** *Oblique shocks* occur when **a gas flowing at supersonic speeds strikes a flat or inclined surface**. Normal shock waves are perpendicular to flow whereas inclined shock waves, as the name implies, are typically **inclined relative to the flow direction**. Also, normal shocks form a straight line whereas **oblique shocks can be straight or curved**, depending on the surface geometry.

***Discussion*** In addition, while a normal shock must go from supersonic (Ma > 1) to subsonic (Ma < 1), the Mach number downstream of an oblique shock can be either supersonic or subsonic.

---

**12-73C**

**Solution** We are to discuss whether the flow upstream and downstream of an oblique shock needs to be supersonic.

***Analysis*** **Yes**, the *upstream flow* has to be supersonic for an oblique shock to occur. **No**, the flow *downstream* of an oblique shock can be subsonic, sonic, and even supersonic.

***Discussion*** The latter is not true for normal shocks. For a normal shock, the flow must always go from supersonic (Ma > 1) to subsonic (Ma < 1).

---

**12-74C**

**Solution** We are to analyze a claim about oblique shock analysis.

***Analysis*** **Yes, the claim is correct**. Conversely, normal shocks can be thought of as special oblique shocks in which the shock angle is $\beta = \pi/2$, or 90°.

***Discussion*** The component of flow in the direction normal to the oblique shock acts exactly like a normal shock. We can think of the flow parallel to the oblique shock as "going along for the ride" – it does not affect anything.

---

**12-75C**

**Solution** We are to discuss shock detachment at the nose of a 2-D wedge-shaped body.

***Analysis*** **When the wedge half-angle $\delta$ is greater than the maximum deflection angle $\theta_{max}$**, the shock becomes curved and detaches from the nose of the wedge, forming what is called a *detached oblique shock* or a *bow wave*. The numerical value of the shock angle at the nose is $\beta = 90^o$.

***Discussion*** When $\delta$ is less than $\theta_{max}$, the oblique shock is attached to the nose.

---

**12-76C**

**Solution** We are to discuss the shock at the nose of a rounded body in supersonic flow.

***Analysis*** When supersonic flow impinges on a blunt body like the rounded nose of an aircraft, the wedge half-angle $\delta$ at the nose is $90^o$, and an attached oblique shock cannot exist, regardless of Mach number. Therefore, **a detached oblique shock must occur in front of *all* such blunt-nosed bodies**, whether two-dimensional, axisymmetric, or fully three-dimensional.

***Discussion*** Since $\delta = 90^o$ at the nose, $\delta$ is always greater than $\theta_{max}$, regardless of Ma or the shape of the rest of the body.

---

**12-77C**

**Solution** We are to discuss the applicability of the isentropic flow relations across shocks and expansion waves.

***Analysis*** The isentropic relations of ideal gases are ***not* applicable for flows across (*a*) normal shock waves and (*b*) oblique shock waves**, but they ***are* applicable for flows across (*c*) Prandtl-Meyer expansion waves**.

***Discussion*** Flow across any kind of shock wave involves irreversible losses – hence, it cannot be isentropic.

---

**12-78**

**Solution** For an ideal gas flowing through a normal shock, a relation for $V_2/V_1$ in terms of $k$, $Ma_1$, and $Ma_2$ is to be developed.

***Analysis*** The conservation of mass relation across the shock is $\rho_1 V_1 = \rho_2 V_2$ and it can be expressed as

$$\frac{V_2}{V_1} = \frac{\rho_1}{\rho_2} = \frac{P_1/RT_1}{P_2/RT_2} = \left(\frac{P_1}{P_2}\right)\left(\frac{T_2}{T_1}\right)$$

From Eqs. 12-35 and 12-38,

$$\frac{V_2}{V_1} = \left(\frac{1+k\text{Ma}_2^2}{1+k\text{Ma}_1^2}\right)\left(\frac{1+\text{Ma}_1^2(k-1)/2}{1+\text{Ma}_2^2(k-1)/2}\right)$$

***Discussion*** This is an important relation as it enables us to determine the velocity ratio across a normal shock when the Mach numbers before and after the shock are known.

---

**12-79**

**Solution** Air flowing through a converging-diverging nozzle experiences a normal shock at the exit. The effect of the shock wave on various properties is to be determined.

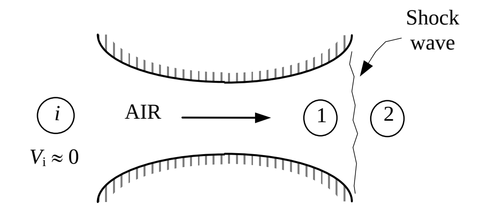
***Assumptions*** **1** Air is an ideal gas with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic before the shock occurs. **3** The shock wave occurs at the exit plane.

***Properties*** The properties of air are $k = 1.4$ and $R = 0.287$ kJ/kg·K.

***Analysis*** The inlet stagnation properties in this case are identical to the inlet properties since the inlet velocity is negligible. Then,

$$P_{01} = P_i = 1 \text{ MPa}$$

$$T_{01} = T_i = 300 \text{ K}$$




Then,

$$T_1 = T_{01}\left(\frac{2}{2+(k-1)\text{Ma}_1^2}\right) = (300\text{ K})\left(\frac{2}{2+(1.4\text{-}1)2^2}\right) = 166.7\text{ K}$$

and

$$P_1 = P_{01}\left(\frac{T_1}{T_0}\right)^{k/(k-1)} = (1\text{ MPa})\left(\frac{166.7}{300}\right)^{1.4/0.4} = 0.1278\text{ MPa}$$

The fluid properties after the shock (denoted by subscript 2) are related to those before the shock through the functions listed in Table A-14. For $\text{Ma}_1 = 2.0$ we read

$$\text{Ma}_2 = \mathbf{0.5774},\quad \frac{P_{02}}{P_{02}} = 0.7209,\quad \frac{P_2}{P_1} = 4.5000,\ \text{and}\ \frac{T_2}{T_1} = 1.6875$$

Then the stagnation pressure $P_{02}$, static pressure $P_2$, and static temperature $T_2$, are determined to be

$$P_{02} = 0.7209P_{01} = (0.7209)(1.0\text{ MPa}) = \mathbf{0.721\ MPa}$$

$$P_2 = 4.5000P_1 = (4.5000)(0.1278\text{ MPa}) = \mathbf{0.575\ MPa}$$

$$T_2 = 1.6875T_1 = (1.6875)(166.7\text{ K}) = \mathbf{281\ K}$$

The air velocity after the shock can be determined from $V_2 = \text{Ma}_2c_2$, where $c_2$ is the speed of sound at the exit conditions after the shock,

$$V_2 = \text{Ma}_2c_2 = \text{Ma}_2\sqrt{kRT_2} = (0.5774)\sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(281\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = \mathbf{194\ \ m/s}$$

***Discussion*** We can also solve this problem using the relations for normal shock functions. The results would be identical.

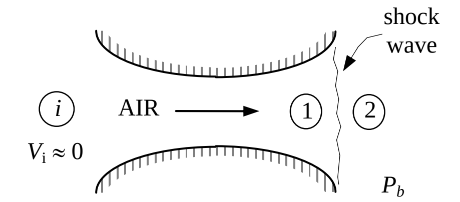
## 12-80

**Solution** Air enters a converging-diverging nozzle at a specified state. The required back pressure that produces a normal shock at the exit plane is to be determined for the specified nozzle geometry.

***Assumptions*** **1** Air is an ideal gas. **2** Flow through the nozzle is steady, one-dimensional, and isentropic before the shock occurs. **3** The shock wave occurs at the exit plane.

***Analysis*** The inlet stagnation pressure in this case is identical to the inlet pressure since the inlet velocity is negligible. Since the flow before the shock to be isentropic,

$$P_{01} = P_i = 2 \text{ MPa}$$




It is specified that $A/A^*$ =3.5. From Table A-13, Mach number and the pressure ratio which corresponds to this area ratio are the $Ma_1$ =2.80 and $P_1/P_{01}$ = 0.0368. The pressure ratio across the shock for this $Ma_1$ value is, from Table A-14, $P_2/P_1$ = 8.98. Thus the back pressure, which is equal to the static pressure at the nozzle exit, must be

$$P_2 = 8.98P_1 = 8.98\times0.0368P_{01} = 8.98\times0.0368\times(2 \text{ MPa}) = \mathbf{0.661\ MPa}$$

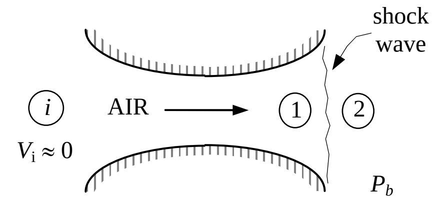
***Discussion*** We can also solve this problem using the relations for compressible flow and normal shock functions. The results would be identical.

---

## 12-81

**Solution** Air enters a converging-diverging nozzle at a specified state. The required back pressure that produces a normal shock at the exit plane is to be determined for the specified nozzle geometry.

***Assumptions*** **1** Air is an ideal gas. **2** Flow through the nozzle is steady, one-dimensional, and isentropic before the shock occurs.




***Analysis*** The inlet stagnation pressure in this case is identical to the inlet pressure since the inlet velocity is negligible. Since the flow before the shock to be isentropic,

$$P_{0x} = P_i = 2 \text{ MPa}$$

It is specified that $A/A^*$ = 2. From Table A-13, the Mach number and the pressure ratio which corresponds to this area ratio are the $Ma_1$ =2.20 and $P_1/P_{01}$ = 0.0935. The pressure ratio across the shock for this $M_1$ value is, from Table A-14, $P_2/P_1$ = 5.48. Thus the back pressure, which is equal to the static pressure at the nozzle exit, must be

$$P_2 = 5.48P_1 = 5.48\times0.0935P_{01} = 5.48\times0.0935\times(2 \text{ MPa}) = \mathbf{1.02\ MPa}$$

***Discussion*** We can also solve this problem using the relations for compressible flow and normal shock functions. The results would be identical.

---

**12-82**

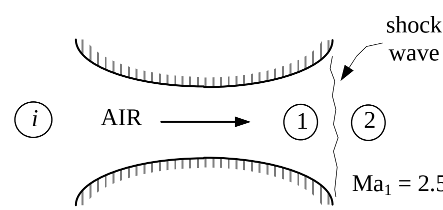
**Solution** Air flowing through a nozzle experiences a normal shock. The effect of the shock wave on various properties is to be determined. Analysis is to be repeated for helium under the same conditions.

***Assumptions*** **1** Air and helium are ideal gases with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic before the shock occurs.

***Properties*** The properties of air are $k = 1.4$ and $R = 0.287$ kJ/kg·K, and the properties of helium are $k = 1.667$ and R = 2.0769 kJ/kg·K.

***Analysis*** The air properties upstream the shock are

$\mathrm{Ma}_1 = 2.5$, $P_1 = 61.64$ kPa, and $T_1 = 262.15$ K




Fluid properties after the shock (denoted by subscript 2) are related to those before the shock through the functions in Table A-14. For $\mathrm{Ma}_1 = 2.5$,

$$\mathrm{Ma}_2 = \mathbf{0.513},\ \frac{P_{02}}{P_1} = 8.5262,\ \frac{P_2}{P_1} = 7.125,\ \text{and}\ \frac{T_2}{T_1} = 2.1375$$

Then the stagnation pressure $P_{02}$, static pressure $P_2$, and static temperature $T_2$, are determined to be

$P_{02} = 8.5261P_1 = (8.5261)(61.64\text{ kPa}) =$ **526 kPa**

$P_2 = 7.125P_1 = (7.125)(61.64\text{ kPa}) =$ **439 kPa**

$T_2 = 2.1375T_1 = (2.1375)(262.15\text{ K}) =$ **560 K**

The air velocity after the shock can be determined from $V_2 = \mathrm{Ma}_2 c_2$, where $c_2$ is the speed of sound at the exit conditions after the shock,

$$V_2 = \mathrm{Ma}_2 c_2 = \mathrm{Ma}_2\sqrt{kRT_2} = (0.513)\sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(560.3\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = \mathbf{243\ m/s}$$

We now repeat the analysis for helium. This time we cannot use the tabulated values in Table A-14 since $k$ is not 1.4. Therefore, we have to calculate the desired quantities using the analytical relations,

$$\mathrm{Ma}_2 = \left(\frac{\mathrm{Ma}_1^2 + 2/(k-1)}{2\mathrm{Ma}_1^2 k/(k-1) - 1}\right)^{1/2} = \left(\frac{2.5^2 + 2/(1.667-1)}{2\times 2.5^2 \times 1.667/(1.667-1) - 1}\right)^{1/2} = \mathbf{0.553}$$

$$\frac{P_2}{P_1} = \frac{1 + k\mathrm{Ma}_1^2}{1 + k\mathrm{Ma}_2^2} = \frac{1 + 1.667\times 2.5^2}{1 + 1.667\times 0.553^2} = 7.5632$$

$$\frac{T_2}{T_1} = \frac{1 + \mathrm{Ma}_1^2(k-1)/2}{1 + \mathrm{Ma}_2^2(k-1)/2} = \frac{1 + 2.5^2(1.667-1)/2}{1 + 0.553^2(1.667-1)/2} = 2.7989$$

$$\frac{P_{02}}{P_1} = \left(\frac{1 + k\mathrm{Ma}_1^2}{1 + k\mathrm{Ma}_2^2}\right)\left(1 + (k-1)\mathrm{Ma}_2^2/2\right)^{k/(k-1)}$$

$$= \left(\frac{1 + 1.667\times 2.5^2}{1 + 1.667\times 0.553^2}\right)\left(1 + (1.667-1)\times 0.553^2/2\right)^{1.667/0.667} = 9.641$$

Thus, $P_{02} = 11.546P_1 = (11.546)(61.64\text{ kPa}) =$ **712 kPa**

$P_2 = 7.5632P_1 = (7.5632)(61.64\text{ kPa}) =$ **466 kPa**

$T_2 = 2.7989T_1 = (2.7989)(262.15\text{ K}) =$ **734 K**

$$V_2 = \mathrm{Ma}_2 c_2 = \mathrm{Ma}_2\sqrt{kRT_y} = (0.553)\sqrt{(1.667)(2.0769\text{ kJ/kg}\cdot\text{K})(733.7\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = \mathbf{881\ m/s}$$

***Discussion*** The velocity and Mach number are higher for helium than for air due to the different values of $k$ and $R$.

**12-83**

**Solution** Air flowing through a nozzle experiences a normal shock. The entropy change of air across the normal shock wave is to be determined.

***Assumptions*** **1** Air and helium are ideal gases with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic before the shock occurs.

***Properties*** The properties of air are $R$ = 0.287 kJ/kg·K and $c_p$ = 1.005 kJ/kg·K, and the properties of helium are $R$ = 2.0769 kJ/kg·K and $c_p$ = 5.1926 kJ/kg·K.

***Analysis*** The entropy change across the shock is determined to be

$$s_2 - s_1 = c_p \ln\frac{T_2}{T_1} - R\ln\frac{P_2}{P_1} = (1.005\ \text{kJ/kg}\cdot\text{K})\ln(2.1375) - (0.287\ \text{kJ/kg}\cdot\text{K})\ln(7.125) = \mathbf{0.200\ \ kJ/kg\cdot K}$$

For helium, the entropy change across the shock is determined to be

$$s_2 - s_1 = c_p \ln\frac{T_2}{T_1} - R\ln\frac{P_2}{P_1} = (5.1926\ \text{kJ/kg}\cdot\text{K})\ln(2.7989) - (2.0769\ \text{kJ/kg}\cdot\text{K})\ln(7.5632) = \mathbf{1.14\ kJ/kg\cdot K}$$

***Discussion*** Note that shock wave is a highly dissipative process, and the entropy generation is large during shock waves.

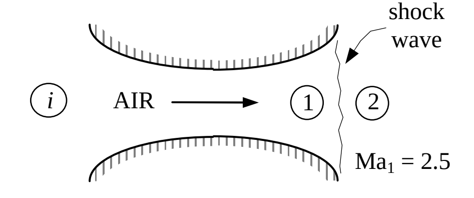
**12-84E** **[*Also solved using EES on enclosed DVD*]**

**Solution** Air flowing through a nozzle experiences a normal shock. Effect of the shock wave on various properties is to be determined. Analysis is to be repeated for helium

***Assumptions*** **1** Air and helium are ideal gases with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic before the shock occurs.

***Properties*** The properties of air are $k$ = 1.4 and $R$ = 0.06855 Btu/lbm·R, and the properties of helium are $k$ = 1.667 and R = 0.4961 Btu/lbm·R.

***Analysis*** The air properties upstream the shock are




$Ma_1 = 2.5$, $P_1$ = 10 psia, and $T_1$ = 440.5 R

Fluid properties after the shock (denoted by subscript 2) are related to those before the shock through the functions listed in Table A-14. For $Ma_1 = 2.5$,

$$\text{Ma}_2 = \mathbf{0.513},\ \frac{P_{02}}{P_1} = 8.5262,\ \frac{P_2}{P_1} = 7.125,\ \text{and}\ \frac{T_2}{T_1} = 2.1375$$

Then the stagnation pressure $P_{02}$, static pressure $P_2$, and static temperature $T_2$, are determined to be

$P_{02} = 8.5262P_1$ = (8.5262)(10 psia) = **85.3 psia**

$P_2 = 7.125P_1$ = (7.125)(10 psia) = **71.3 psia**

$T_2 = 2.1375T_1$ = (2.1375)(440.5 R) = **942 R**

The air velocity after the shock can be determined from $V_2 = \text{Ma}_2c_2$, where $c_2$ is the speed of sound at the exit conditions after the shock,

$$V_2 = \text{Ma}_2 c_2 = \text{Ma}_2\sqrt{kRT_2} = (0.513)\sqrt{(1.4)(0.06855\ \text{Btu/1bm}\cdot\text{R})(941.6\ \text{R})\left(\frac{25{,}037\ \text{ft}^2/\text{s}^2}{1\ \text{Btu/1bm}}\right)} = \mathbf{772\ \ ft/s}$$

We now repeat the analysis for helium. This time we cannot use the tabulated values in Table A-14 since $k$ is not 1.4. Therefore, we have to calculate the desired quantities using the analytical relations,

$$\text{Ma}_2 = \left(\frac{\text{Ma}_1^2 + 2/(k-1)}{2\text{Ma}_1^2 k/(k-1) - 1}\right)^{1/2} = \left(\frac{2.5^2 + 2/(1.667-1)}{2\times 2.5^2 \times 1.667/(1.667-1) - 1}\right)^{1/2} = \mathbf{0.553}$$

$$\frac{P_2}{P_1} = \frac{1+k\text{Ma}_1^2}{1+k\text{Ma}_2^2} = \frac{1+1.667\times 2.5^2}{1+1.667\times 0.553^2} = 7.5632$$

$$\frac{T_2}{T_1} = \frac{1+\text{Ma}_1^2(k-1)/2}{1+\text{Ma}_2^2(k-1)/2} = \frac{1+2.5^2(1.667-1)/2}{1+0.553^2(1.667-1)/2} = 2.7989$$

$$\frac{P_{02}}{P_1} = \left(\frac{1+k\text{Ma}_1^2}{1+k\text{Ma}_2^2}\right)\left(1+(k-1)\text{Ma}_2^2/2\right)^{k/(k-1)}$$

$$= \left(\frac{1+1.667\times 2.5^2}{1+1.667\times 0.553^2}\right)\left(1+(1.667-1)\times 0.553^2/2\right)^{1.667/0.667} = 9.641$$

Thus, $P_{02} = 11.546P_1$ = (11.546)(10 psia) = **115 psia**

$P_2 = 7.5632P_1$ = (7.5632)(10 psia) = **75.6 psia**

$T_2 = 2.7989T_1$ = (2.7989)(440.5 R) = **1233 R**

$$V_2 = \text{Ma}_2 c_2 = \text{Ma}_2\sqrt{kRT_2} = (0.553)\sqrt{(1.667)(0.4961\ \text{Btu/1bm.R})(1232.9\ \text{R})\left(\frac{25{,}037\ \text{ft}^2/\text{s}^2}{1\ \text{Btu/1bm}}\right)} = \mathbf{2794\ \ ft/s}$$

***Discussion*** This problem could also be solved using the relations for compressible flow and normal shock functions. The results would be identical.


**12-85E** 

**Solution** We are to reconsider Prob. 12-84E. Using EES (or other) software, we are to study the effects of both air and helium flowing steadily in a nozzle when there is a normal shock at a Mach number in the range 2 < Mx < 3.5. In addition to the required information, we are to calculate the entropy change of the air and helium across the normal shock, and tabulate the results in a parametric table.

***Analysis*** We use EES to calculate the entropy change of the air and helium across the normal shock. The results are given in the Parametric Table for 2 < M_x < 3.5.

```
Procedure NormalShock(M_x,k:M_y,PyOPx, TyOTx,RhoyORhox, PoyOPox, PoyOPx)
    If M_x < 1 Then
            M_y = -1000;PyOPx=-1000;TyOTx=-1000;RhoyORhox=-1000
            PoyOPox=-1000;PoyOPx=-1000
    else
            M_y=sqrt( (M_x^2+2/(k-1)) / (2*M_x^2*k/(k-1)-1) )
            PyOPx=(1+k*M_x^2)/(1+k*M_y^2)
            TyOTx=( 1+M_x^2*(k-1)/2  )/(1+M_y^2*(k-1)/2  )
            RhoyORhox=PyOPx/TyOTx
            PoyOPox=M_x/M_y*(  (1+M_y^2*(k-1)/2)/ (1+M_x^2*(k-1)/2)   )^((k+1)/(2*(k-1)))
            PoyOPx=(1+k*M_x^2)*(1+M_y^2*(k-1)/2)^(k/(k-1))/(1+k*M_y^2)
    Endif
End

Function ExitPress(P_back,P_crit)
If P_back>=P_crit then ExitPress:=P_back     "Unchoked Flow Condition"
If P_back<P_crit then ExitPress:=P_crit       "Choked Flow Condition"
End

Procedure GetProp(Gas$:Cp,k,R) "Cp and k data are from Text Table A.2E"
   M=MOLARMASS(Gas$)            "Molar mass of  Gas$"
   R= 1545/M                    "Particular gas constant for Gas$, ft-lbf/lbm-R"
                                "k = Ratio of Cp to Cv"
                                "Cp = Specific heat at constant pressure"
   if Gas$='Air' then
                                Cp=0.24"Btu/lbm-R"; k=1.4
   endif
   if Gas$='CO2' then
                                Cp=0.203"Btu/lbm_R"; k=1.289
   endif
   if Gas$='Helium' then
                                Cp=1.25"Btu/lbm-R"; k=1.667
   endif
End

"Variable Definitions:"
"M = flow Mach Number"
"P_ratio = P/P_o  for compressible, isentropic flow"
"T_ratio = T/T_o for compressible, isentropic flow"
"Rho_ratio= Rho/Rho_o for compressible, isentropic flow"
"A_ratio=A/A* for compressible, isentropic flow"
"Fluid properties before the shock are denoted with a subscript x"
"Fluid properties after the shock are denoted with a subscript y"
"M_y = Mach Number down stream of normal shock"
"PyOverPx= P_y/P_x  Pressue ratio across normal shock"
"TyOverTx =T_y/T_x  Temperature ratio across normal shock"
"RhoyOverRhox=Rho_y/Rho_x Density ratio across normal shock"
"PoyOverPox = P_oy/P_ox  Stagantion pressure ratio across normal shock"
"PoyOverPx = P_oy/P_x  Stagnation pressure after normal shock ratioed to pressure before shock"

"Input Data"
{P_x = 10 "psia"}               "Values of P_x, T_x, and M_x are set in the Parametric Table"
{T_x = 440.5 "R"}
{M_x = 2.5}
```

```
Gas$='Air'  "This program has been written for the gases Air, CO2, and Helium"
Call GetProp(Gas$:Cp,k,R)
Call NormalShock(M_x,k:M_y,PyOverPx, TyOverTx,RhoyOverRhox, PoyOverPox, PoyOverPx)
P_oy_air=P_x*PoyOverPx          "Stagnation pressure after the shock"
P_y_air=P_x*PyOverPx            "Pressure after the shock"
T_y_air=T_x*TyOverTx            "Temperature after the shock"
M_y_air=M_y                     "Mach number after the shock"

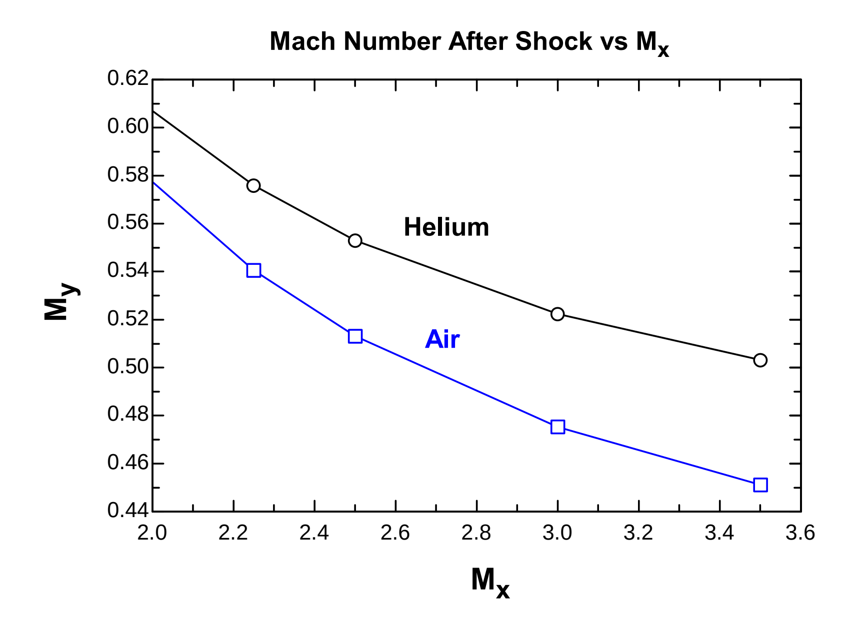
"The velocity after the shock can be found from the product of the Mach number and
speed of sound after the shock."
C_y_air = sqrt(k*R"ft-lbf/lbm_R"*T_y_air"R"*32.2 "lbm-ft/lbf-s^2")
V_y_air=M_y_air*C_y_air
DELTAs_air=entropy(air,T=T_y_air, P=P_y_air) -entropy(air,T=T_x,P=P_x)

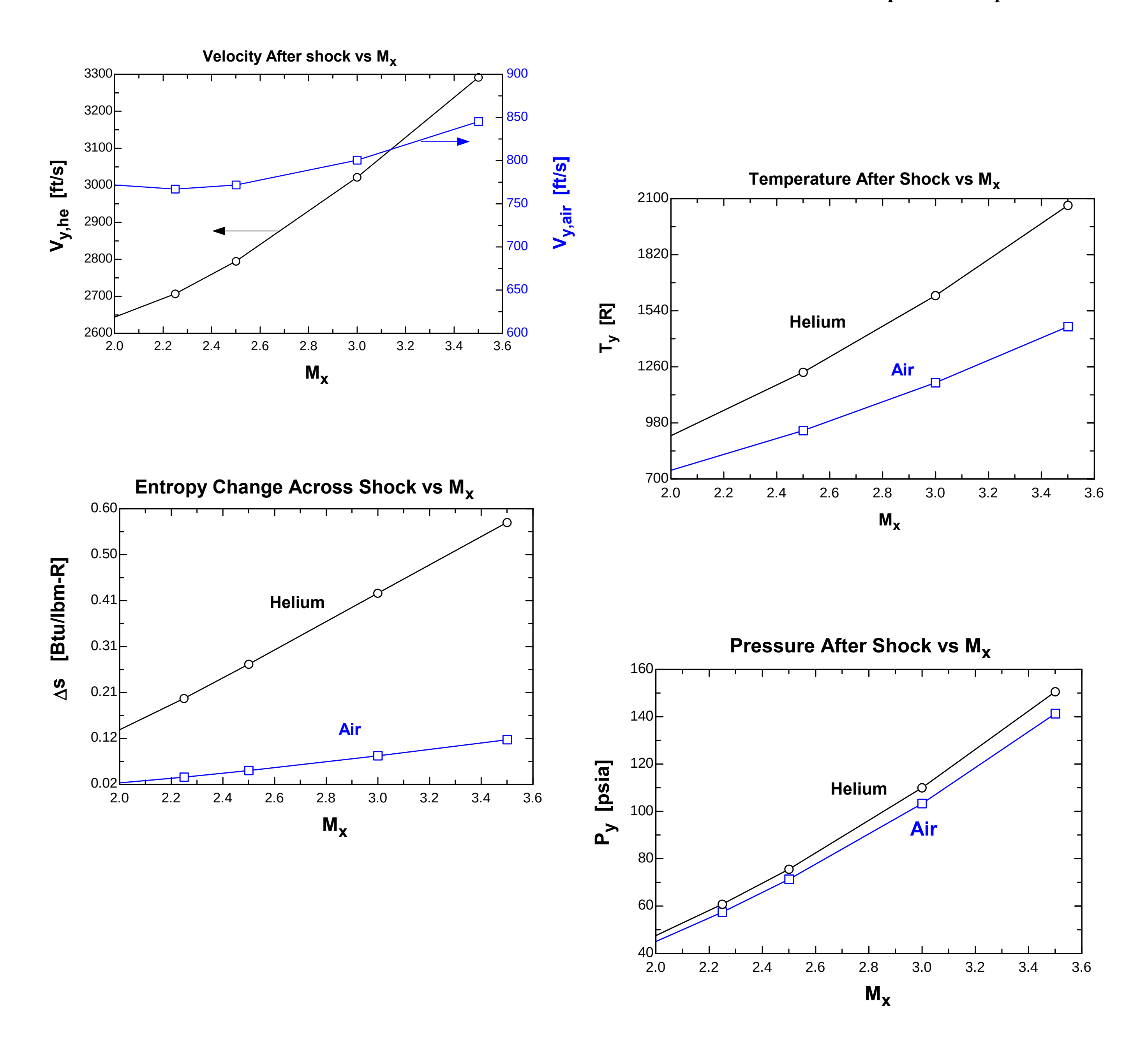
Gas2$='Helium'  "Gas2$ can be either Helium or CO2"
Call GetProp(Gas2$:Cp_2,k_2,R_2)
Call NormalShock(M_x,k_2:M_y2,PyOverPx2, TyOverTx2,RhoyOverRhox2, PoyOverPox2, PoyOverPx2)
P_oy_he=P_x*PoyOverPx2          "Stagnation pressure after the shock"
P_y_he=P_x*PyOverPx2            "Pressure after the shock"
T_y_he=T_x*TyOverTx2            "Temperature after the shock"
M_y_he=M_y2                     "Mach number after the shock"

"The velocity after the shock can be found from the product of the Mach number and
speed of sound after the shock."
C_y_he = sqrt(k_2*R_2"ft-lbf/lbm_R"*T_y_he"R"*32.2 "lbm-ft/lbf-s^2")
V_y_he=M_y_he*C_y_he
DELTAs_he=entropy(helium,T=T_y_he, P=P_y_he) -entropy(helium,T=T_x,P=P_x)
```

The parametric table and the corresponding plots are shown below.

| $V_{y,he}$ [ft/s] | $V_{y,air}$ [ft/s] | $T_{y,he}$ [R] | $T_{y,air}$ [R] | $T_x$ [R] | $P_{y,he}$ [psia] | $P_{y,air}$ [psia] | $P_x$ [psia] | $P_{oy,he}$ [psia] | $P_{oy,air}$ [psia] | $M_{y,he}$ | $M_{y,air}$ | $M_x$ | $\Delta s_{he}$ [Btu/lbm-R] | $\Delta s_{air}$ [Btu/lbm-R] |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| 2644 | 771.9 | 915.6 | 743.3 | 440.5 | 47.5 | 45 | 10 | 63.46 | 56.4 | 0.607 | 0.5774 | 2 | 0.1345 | 0.0228 |
| 2707 | 767.1 | 1066 | 837.6 | 440.5 | 60.79 | 57.4 | 10 | 79.01 | 70.02 | 0.5759 | 0.5406 | 2.25 | 0.2011 | 0.0351 |
| 2795 | 771.9 | 1233 | 941.6 | 440.5 | 75.63 | 71.25 | 10 | 96.41 | 85.26 | 0.553 | 0.513 | 2.5 | 0.2728 | 0.04899 |
| 3022 | 800.4 | 1616 | 1180 | 440.5 | 110 | 103.3 | 10 | 136.7 | 120.6 | 0.5223 | 0.4752 | 3 | 0.4223 | 0.08 |
| 3292 | 845.4 | 2066 | 1460 | 440.5 | 150.6 | 141.3 | 10 | 184.5 | 162.4 | 0.5032 | 0.4512 | 3.5 | 0.5711 | 0.1136 |






***Discussion*** In all cases, regardless of the fluid or the Mach number, entropy increases across a shock wave. This is because a shock wave involves irreversibilities.

**12-86**

**Solution** Air flowing through a nozzle experiences a normal shock. Various properties are to be calculated before and after the shock.

***Assumptions*** **1** Air is an ideal gas with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic before the shock occurs.

***Properties*** The properties of air at room temperature are $k$= 1.4, $R$ = 0.287 kJ/kg·K, and $c_p$ = 1.005 kJ/kg·K.

***Analysis*** The stagnation temperature and pressure before the shock are

$$T_{01} = T_1 + \frac{V_1^2}{2c_p} = 217 + \frac{(680\text{ m/s})^2}{2(1.005\text{ kJ/kg}\cdot\text{K})}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 447.0\text{ K}$$

$$P_{01} = P_1\left(\frac{T_{01}}{T_1}\right)^{k/(k-1)} = (22.6\text{ kPa})\left(\frac{447.0\text{ K}}{217\text{ K}}\right)^{1.4/(1.4-1)} = 283.6\text{ kPa}$$



The velocity and the Mach number before the shock are determined from

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(217.0\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = \mathbf{295.3\ m/s}$$

and

$$\text{Ma}_1 = \frac{V_1}{c_1} = \frac{680\text{ m/s}}{295.3\text{ m/s}} = \mathbf{2.30}$$

The fluid properties after the shock (denoted by subscript y) are related to those before the shock through the functions listed in Table A-14. For $\text{Ma}_1$ = 2.30 we read

$$\text{Ma}_2 = 0.5344 \cong \mathbf{0.534}, \quad \frac{P_{02}}{P_1} = 7.2937, \quad \frac{P_2}{P_1} = 6.005, \quad \text{and} \quad \frac{T_2}{T_1} = 1.9468$$

Then the stagnation pressure $P_{02}$ , static pressure $P_2$ , and static temperature $T_2$ , are determined to be

$P_{02}$ = 7.2937$P_1$ = (7.2937)(22.6 kPa) = **165 kPa**

$P_2$ = 6.005$P_1$ = (6.005)(22.6 kPa) = **136 kPa**

$T_2$ = 1.9468$T_1$ = (1.9468)(217 K) = **423 K**

The air velocity after the shock can be determined from $V_2 = \text{Ma}_2 c_2$, where $c_2$ is the speed of sound at the exit conditions after the shock,

$$V_2 = \text{Ma}_2 c_2 = \text{Ma}_2\sqrt{kRT_2} = (0.5344)\sqrt{(1.4)(0.287\text{ kJ/kg.K})(422.5\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = \mathbf{220\ m/s}$$

***Discussion*** This problem could also be solved using the relations for compressible flow and normal shock functions. The results would be identical.

---

**12-87**

**Solution** Air flowing through a nozzle experiences a normal shock. The entropy change of air across the normal shock wave is to be determined.

***Assumptions*** **1** Air is an ideal gas with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic before the shock occurs.

***Properties*** The properties of air at room temperature are $R$ = 0.287 kJ/kg·K and $c_p$ = 1.005 kJ/kg·K.

***Analysis*** The entropy change across the shock is determined to be

$$s_2 - s_1 = c_p \ln\frac{T_2}{T_1} - R\ln\frac{P_2}{P_1} = (1.005\text{ kJ/kg}\cdot\text{K})\ln(1.9468) - (0.287\text{ kJ/kg}\cdot\text{K})\ln(6.005) = \mathbf{0.155\ \frac{kJ}{kg\cdot K}}$$

***Discussion*** A shock wave is a highly dissipative process, and the entropy generation is large during shock waves.

---


**12-88** 

**Solution** The entropy change of air across the shock for upstream Mach numbers between 0.5 and 1.5 is to be determined and plotted.

***Assumptions*** **1** Air is an ideal gas. **2** Flow through the nozzle is steady, one-dimensional, and isentropic before the shock occurs.

***Properties*** The properties of air are $k$ = 1.4, $R$ = 0.287 kJ/kg·K, and $c_p$ = 1.005 kJ/kg·K.

***Analysis*** The entropy change across the shock is determined to be

$$s_2 - s_1 = c_p \ln\frac{T_2}{T_1} - R\ln\frac{P_2}{P_1}$$

where

$$\text{Ma}_2 = \left(\frac{\text{Ma}_1^2 + 2/(k-1)}{2\text{Ma}_1^2 k/(k-1) - 1}\right)^{1/2}, \quad \frac{P_2}{P_1} = \frac{1+k\text{Ma}_1^2}{1+k\text{Ma}_2^2}, \text{ and } \frac{T_2}{T_1} = \frac{1+\text{Ma}_1^2(k-1)/2}{1+\text{Ma}_2^2(k-1)/2}$$

The results of the calculations can be tabulated as

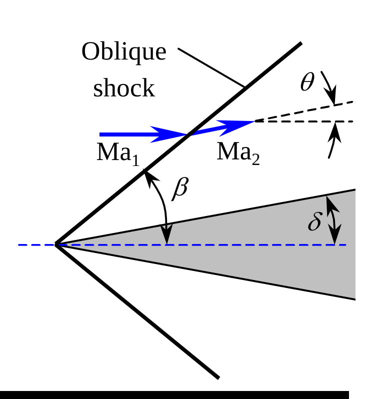
| $\text{Ma}_1$ | $\text{Ma}_2$ | $T_2/T_1$ | $P_2/P_1$ | $s_2 - s_1$ |
|---|---|---|---|---|
| 0.5 | 2.6458 | 0.1250 | 0.4375 | -1.853 |
| 0.6 | 1.8778 | 0.2533 | 0.6287 | -1.247 |
| 0.7 | 1.5031 | 0.4050 | 0.7563 | -0.828 |
| 0.8 | 1.2731 | 0.5800 | 0.8519 | -0.501 |
| 0.9 | 1.1154 | 0.7783 | 0.9305 | -0.231 |
| 1.0 | 1.0000 | 1.0000 | 1.0000 | 0.0 |
| 1.1 | 0.9118 | 1.0649 | 1.2450 | 0.0003 |
| 1.2 | 0.8422 | 1.1280 | 1.5133 | 0.0021 |
| 1.3 | 0.7860 | 1.1909 | 1.8050 | 0.0061 |
| 1.4 | 0.7397 | 1.2547 | 2.1200 | 0.0124 |
| 1.5 | 0.7011 | 1.3202 | 2.4583 | 0.0210 |

***Discussion*** The total entropy change is negative for upstream Mach numbers $\text{Ma}_1$ less than unity. Therefore, normal shocks cannot occur when $\text{Ma}_1 < 1$.

---

**12-89**

**Solution** Supersonic airflow approaches the nose of a two-dimensional wedge and undergoes a straight oblique shock. For a specified Mach number, the minimum shock angle and the maximum deflection angle are to be determined.




***Assumptions*** Air is an ideal gas with a constant specific heat ratio of $k$ = 1.4 (so that Fig. 12-41 is applicable).

***Analysis*** For Ma = 5, we read from Fig. 12-41

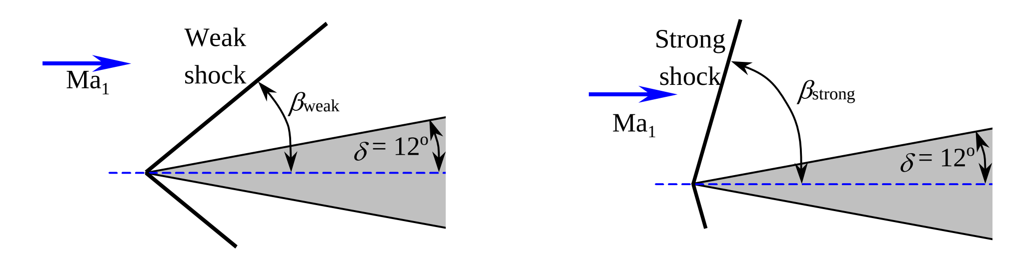
Minimum shock (or wave) angle: $\beta_{min} = \mathbf{12°}$

Maximum deflection (or turning) angle: $\theta_{max} = \mathbf{41.5°}$

***Discussion*** Note that the minimum shock angle decreases and the maximum deflection angle increases with increasing Mach number $\text{Ma}_1$.

**12-90**

**Solution** Air flowing at a specified supersonic Mach number impinges on a two-dimensional wedge, The shock angle, Mach number, and pressure downstream of the weak and strong oblique shock formed by a wedge are to be determined.




***Assumptions*** **1** The flow is steady. **2** The boundary layer on the wedge is very thin. **3** Air is an ideal gas with constant specific heats.

***Properties*** The specific heat ratio of air is $k$ = 1.4.

***Analysis*** On the basis of Assumption #2, we take the deflection angle as equal to the wedge half-angle, i.e., $\theta \approx \delta = 12^o$. Then the two values of oblique shock angle $\beta$ are determined from

$$\tan\theta = \frac{2(\text{Ma}_1^2 \sin^2\beta - 1)/\tan\beta}{\text{Ma}_1^2(k + \cos 2\beta) + 2} \quad \rightarrow \quad \tan 12° = \frac{2(3.4^2 \sin^2\beta - 1)/\tan\beta}{3.4^2(1.4 + \cos 2\beta) + 2}$$

which is implicit in $\beta$. Therefore, we solve it by an iterative approach or with an equation solver such as EES. It gives $\boldsymbol{\beta_{weak}}$ **= 26.75°** and $\boldsymbol{\beta_{strong}}$ **=86.11°**. Then the upstream "normal" Mach number $Ma_{1,n}$ becomes

*Weak shock*: $\text{Ma}_{1,n} = \text{Ma}_1 \sin\beta = 3.4 \sin 26.75° = 1.531$

*Strong shock*: $\text{Ma}_{1,n} = \text{Ma}_1 \sin\beta = 3.4 \sin 86.11° = 3.392$

Also, the downstream normal Mach numbers $Ma_{2,n}$ become

*Weak shock*: $$\text{Ma}_{2,n} = \sqrt{\frac{(k-1)\text{Ma}_{1,n}^2 + 2}{2k\text{Ma}_{1,n}^2 - k + 1}} = \sqrt{\frac{(1.4-1)(1.531)^2 + 2}{2(1.4)(1.531)^2 - 1.4 + 1}} = 0.6905$$

*Strong shock*: $$\text{Ma}_{2,n} = \sqrt{\frac{(k-1)\text{Ma}_{1,n}^2 + 2}{2k\text{Ma}_{1,n}^2 - k + 1}} = \sqrt{\frac{(1.4-1)(3.392)^2 + 2}{2(1.4)(3.392)^2 - 1.4 + 1}} = 0.4555$$

The downstream pressure for each case is determined to be

*Weak shock*: $$P_2 = P_1 \frac{2k\text{Ma}_{1,n}^2 - k + 1}{k+1} = (60 \text{ kPa}) \frac{2(1.4)(1.531)^2 - 1.4 + 1}{1.4 + 1} = \mathbf{154 \text{ kPa}}$$

*Strong shock*: $$P_2 = P_1 \frac{2k\text{Ma}_{1,n}^2 - k + 1}{k+1} = (60 \text{ kPa}) \frac{2(1.4)(3.392)^2 - 1.4 + 1}{1.4 + 1} = \mathbf{796 \text{ kPa}}$$

The downstream Mach number is determined to be

*Weak shock*: $$\text{Ma}_2 = \frac{\text{Ma}_{2,n}}{\sin(\beta - \theta)} = \frac{0.6905}{\sin(26.75° - 12°)} = \mathbf{2.71}$$

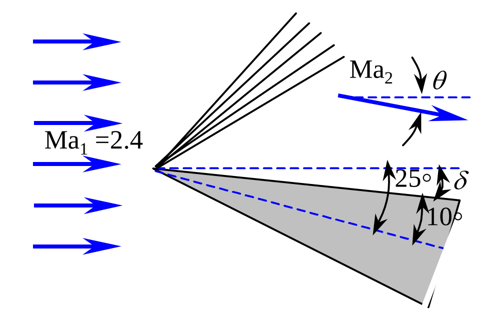
*Strong shock*: $$\text{Ma}_2 = \frac{\text{Ma}_{2,n}}{\sin(\beta - \theta)} = \frac{0.4555}{\sin(86.11° - 12°)} = \mathbf{0.474}$$

***Discussion*** Note that the change in Mach number and pressure across the *strong shock* are much greater than the changes across the *weak shock*, as expected. For both the weak and strong oblique shock cases, $Ma_{1,n}$ is supersonic and $Ma_{2,n}$ is subsonic. However, $Ma_2$ is *supersonic* across the weak oblique shock, but *subsonic* across the strong oblique shock.

**12-91**

**Solution** Air flowing at a specified supersonic Mach number undergoes an expansion turn over a tilted wedge. The Mach number, pressure, and temperature downstream of the sudden expansion above the wedge are to be determined.

***Assumptions*** **1** The flow is steady. **2** The boundary layer on the wedge is very thin. **3** Air is an ideal gas with constant specific heats.




***Properties*** The specific heat ratio of air is $k$ = 1.4.

***Analysis*** On the basis of Assumption #2, the deflection angle is determined to be $\theta \approx \delta$ = 25° - 10° = 15°. Then the upstream and downstream Prandtl-Meyer functions are determined to be

$$\nu(\mathrm{Ma}) = \sqrt{\frac{k+1}{k-1}}\tan^{-1}\left(\sqrt{\frac{k-1}{k+1}(\mathrm{Ma}^2-1)}\right) - \tan^{-1}\left(\sqrt{\mathrm{Ma}^2-1}\right)$$

*Upstream*:

$$\nu(\mathrm{Ma}_1) = \sqrt{\frac{1.4+1}{1.4-1}}\tan^{-1}\left(\sqrt{\frac{1.4-1}{1.4+1}(2.4^2-1)}\right) - \tan^{-1}\left(\sqrt{2.4^2-1}\right) = 36.75°$$

Then the downstream Prandtl-Meyer function becomes

$$\nu(\mathrm{Ma}_2) = \theta + \nu(\mathrm{Ma}_1) = 15° + 36.75° = 51.75°$$

Now $Ma_2$ is found from the Prandtl-Meyer relation, which is now implicit:

*Downstream*: $\nu(\mathrm{Ma}_2) = \sqrt{\frac{1.4+1}{1.4-1}}\tan^{-1}\left(\sqrt{\frac{1.4-1}{1.4+1}(\mathrm{Ma}_2^2-1)}\right) - \tan^{-1}\left(\sqrt{\mathrm{Ma}_2^2-1}\right) = 51.75°$

It gives $Ma_2$ = **3.105**. Then the downstream pressure and temperature are determined from the isentropic flow relations

$$P_2 = \frac{P_2/P_0}{P_1/P_0}P_1 = \frac{[1+\mathrm{Ma}_2^2(k-1)/2]^{-k/(k-1)}}{[1+\mathrm{Ma}_1^2(k-1)/2]^{-k/(k-1)}}P_1 = \frac{[1+3.105^2(1.4-1)/2]^{-1.4/0.4}}{[1+2.4^2(1.4-1)/2]^{-1.4/0.4}}(70\text{ kPa}) = \mathbf{23.8\ kPa}$$

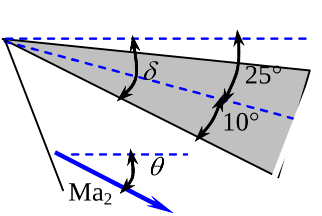
$$T_2 = \frac{T_2/T_0}{T_1/T_0}T_1 = \frac{[1+\mathrm{Ma}_2^2(k-1)/2]^{-1}}{[1+\mathrm{Ma}_1^2(k-1)/2]^{-1}}T_1 = \frac{[1+3.105^2(1.4-1)/2]^{-1}}{[1+2.4^2(1.4-1)/2]^{-1}}(260\text{ K}) = \mathbf{191\ K}$$

Note that this is an expansion, and Mach number increases while pressure and temperature decrease, as expected.

***Discussion*** There are compressible flow calculators on the Internet that solve these implicit equations that arise in the analysis of compressible flow, along with both normal and oblique shock equations; e.g., see www.aoe.vt.edu/~devenpor/aoe3114/calc.html .

**12-92**

**Solution** Air flowing at a specified supersonic Mach number undergoes a compression turn (an oblique shock) over a tilted wedge. The Mach number, pressure, and temperature downstream of the shock below the wedge are to be determined.




***Assumptions*** **1** The flow is steady. **2** The boundary layer on the wedge is very thin. **3** Air is an ideal gas with constant specific heats.

***Properties*** The specific heat ratio of air is $k = 1.4$.

***Analysis*** On the basis of Assumption #2, the deflection angle is determined to be $\theta \approx \delta = 25° + 10° = 35°$. Then the two values of oblique shock angle $\beta$ are determined from

$$\tan\theta = \frac{2(\text{Ma}_1^2 \sin^2\beta - 1)/\tan\beta}{\text{Ma}_1^2(k + \cos 2\beta) + 2} \quad \rightarrow \quad \tan 12° = \frac{2(3.4^2 \sin^2\beta - 1)/\tan\beta}{3.4^2(1.4 + \cos 2\beta) + 2}$$

which is implicit in $\beta$. Therefore, we solve it by an iterative approach or with an equation solver such as EES. It gives $\beta_{weak}$ = 49.86° and $\beta_{strong}$ = 77.66°. Then for the case of strong oblique shock, the upstream "normal" Mach number $Ma_{1,n}$ becomes

$$\text{Ma}_{1,n} = \text{Ma}_1 \sin\beta = 5\sin 77.66° = 4.884$$

Also, the downstream normal Mach numbers $Ma_{2,n}$ become

$$\text{Ma}_{2,n} = \sqrt{\frac{(k-1)\text{Ma}_{1,n}^2 + 2}{2k\text{Ma}_{1,n}^2 - k + 1}} = \sqrt{\frac{(1.4-1)(4.884)^2 + 2}{2(1.4)(4.884)^2 - 1.4 + 1}} = 0.4169$$

The downstream pressure and temperature are determined to be

$$P_2 = P_1 \frac{2k\text{Ma}_{1,n}^2 - k + 1}{k+1} = (70 \text{ kPa})\frac{2(1.4)(4.884)^2 - 1.4 + 1}{1.4 + 1} = \mathbf{1940\ kPa}$$

$$T_2 = T_1 \frac{P_2}{P_1}\frac{\rho_1}{\rho_2} = T_1 \frac{P_2}{P_1}\frac{2 + (k-1)\text{Ma}_{1,n}^2}{(k+1)\text{Ma}_{1,n}^2} = (260 \text{ K})\frac{1940 \text{ kPia}}{70 \text{ kPa}}\frac{2 + (1.4-1)(4.884)^2}{(1.4+1)(4.884)^2} = \mathbf{1450\ K}$$

The downstream Mach number is determined to be

$$\text{Ma}_2 = \frac{\text{Ma}_{2,n}}{\sin(\beta - \theta)} = \frac{0.4169}{\sin(77.66° - 35°)} = \mathbf{0.615}$$

***Discussion*** Note that $Ma_{1,n}$ is supersonic and $Ma_{2,n}$ and $Ma_2$ are subsonic. Also note the huge rise in temperature and pressure across the strong oblique shock, and the challenges they present for spacecraft during reentering the earth's atmosphere.

---

**12-93E**

**Solution** Air flowing at a specified supersonic Mach number is forced to turn upward by a ramp, and weak oblique shock forms. The wave angle, Mach number, pressure, and temperature after the shock are to be determined.

***Assumptions*** **1** The flow is steady. **2** The boundary layer on the wedge is very thin. **3** Air is an ideal gas with constant specific heats.



***Properties*** The specific heat ratio of air is $k$ = 1.4.

***Analysis*** On the basis of Assumption #2, we take the deflection angle as equal to the ramp, i.e., $\theta \approx \delta = 8^{\circ}$. Then the two values of oblique shock angle $\beta$ are determined from

$$\tan\theta = \frac{2(\mathrm{Ma}_1^2 \sin^2\beta - 1)/\tan\beta}{\mathrm{Ma}_1^2(k + \cos 2\beta) + 2} \qquad \rightarrow \qquad \tan 8^{\circ} = \frac{2(2^2 \sin^2\beta - 1)/\tan\beta}{2^2(1.4 + \cos 2\beta) + 2}$$

which is implicit in $\beta$. Therefore, we solve it by an iterative approach or with an equation solver such as EES. It gives $\boldsymbol{\beta_{weak}}$ **= 37.21°** and $\beta_{strong} = 85.05^{\circ}$. Then for the case of weak oblique shock, the upstream "normal" Mach number $\mathrm{Ma}_{1,n}$ becomes



$$\mathrm{Ma}_{1,n} = \mathrm{Ma}_1 \sin\beta = 2\sin 37.21^{\circ} = 1.209$$

Also, the downstream normal Mach numbers $\mathrm{Ma}_{2,n}$ become

$$\mathrm{Ma}_{2,n} = \sqrt{\frac{(k-1)\mathrm{Ma}_{1,n}^2 + 2}{2k\mathrm{Ma}_{1,n}^2 - k + 1}} = \sqrt{\frac{(1.4-1)(1.209)^2 + 2}{2(1.4)(1.209)^2 - 1.4 + 1}} = 0.8363$$

The downstream pressure and temperature are determined to be

$$P_2 = P_1 \frac{2k\mathrm{Ma}_{1,n}^2 - k + 1}{k+1} = (8 \text{ psia}) \frac{2(1.4)(1.209)^2 - 1.4 + 1}{1.4 + 1} = \mathbf{12.3 \text{ psia}}$$

$$T_2 = T_1 \frac{P_2}{P_1}\frac{\rho_1}{\rho_2} = T_1 \frac{P_2}{P_1}\frac{2 + (k-1)\mathrm{Ma}_{1,n}^2}{(k+1)\mathrm{Ma}_{1,n}^2} = (480 \text{ R}) \frac{12.3 \text{ psia}}{8 \text{ psia}} \frac{2 + (1.4-1)(1.209)^2}{(1.4+1)(1.209)^2} = \mathbf{544 \text{ R}}$$

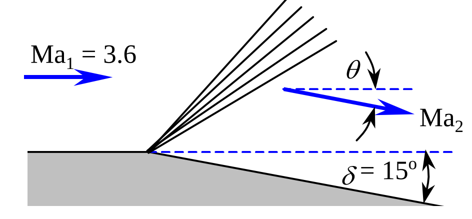
The downstream Mach number is determined to be

$$\mathrm{Ma}_2 = \frac{\mathrm{Ma}_{2,n}}{\sin(\beta - \theta)} = \frac{0.8363}{\sin(37.21^{\circ} - 8^{\circ})} = \mathbf{1.71}$$

***Discussion*** Note that $\mathrm{Ma}_{1,n}$ is supersonic and $\mathrm{Ma}_{2,n}$ is subsonic. However, $\mathrm{Ma}_2$ is *supersonic* across the weak oblique shock (it is *subsonic* across the strong oblique shock).

**12-94**

**Solution** Air flowing at a specified supersonic Mach number undergoes an expansion turn. The Mach number, pressure, and temperature downstream of the sudden expansion along a wall are to be determined.




***Assumptions*** **1** The flow is steady. **2** The boundary layer on the wedge is very thin. **3** Air is an ideal gas with constant specific heats.

***Properties*** The specific heat ratio of air is $k = 1.4$.

***Analysis*** On the basis of Assumption #2, we take the deflection angle as equal to the wedge half-angle, i.e., $\theta \approx \delta = 15°$. Then the upstream and downstream Prandtl-Meyer functions are determined to be

$$\nu(\text{Ma}) = \sqrt{\frac{k+1}{k-1}}\tan^{-1}\left(\sqrt{\frac{k-1}{k+1}(\text{Ma}^2-1)}\right) - \tan^{-1}\left(\sqrt{\text{Ma}^2-1}\right)$$

*Upstream*:

$$\nu(\text{Ma}_1) = \sqrt{\frac{1.4+1}{1.4-1}}\tan^{-1}\left(\sqrt{\frac{1.4-1}{1.4+1}(3.6^2-1)}\right) - \tan^{-1}\left(\sqrt{3.6^2-1}\right) = 60.09°$$

Then the downstream Prandtl-Meyer function becomes

$$\nu(\text{Ma}_2) = \theta + \nu(\text{Ma}_1) = 15° + 60.09° = 75.09°$$

$Ma_2$ is found from the Prandtl-Meyer relation, which is now implicit:

*Downstream*:
$$\nu(\text{Ma}_2) = \sqrt{\frac{1.4+1}{1.4-1}}\tan^{-1}\left(\sqrt{\frac{1.4-1}{1.4+1}\text{Ma}_2^2-1)}\right) - \tan^{-1}\left(\sqrt{\text{Ma}_2^2-1}\right) = 75.09°$$

Solution of this implicit equation gives $Ma_2$ = **4.81**. Then the downstream pressure and temperature are determined from the isentropic flow relations:

$$P_2 = \frac{P_2/P_0}{P_1/P_0}P_1 = \frac{[1+\text{Ma}_2^2(k-1)/2]^{-k/(k-1)}}{[1+\text{Ma}_1^2(k-1)/2]^{-k/(k-1)}}P_1 = \frac{[1+4.81^2(1.4-1)/2]^{-1.4/0.4}}{[1+3.6^2(1.4-1)/2]^{-1.4/0.4}}(40\text{ kPa}) = \mathbf{8.31\ kPa}$$

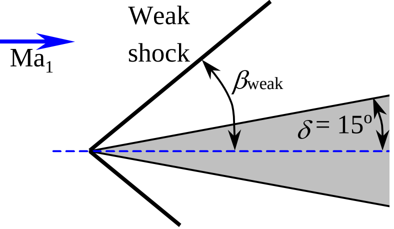
$$T_2 = \frac{T_2/T_0}{T_1/T_0}T_1 = \frac{[1+\text{Ma}_2^2(k-1)/2]^{-1}}{[1+\text{Ma}_1^2(k-1)/2]^{-1}}T_1 = \frac{[1+4.81^2(1.4-1)/2]^{-1}}{[1+3.6^2(1.4-1)/2]^{-1}}(280\text{ K}) = \mathbf{179\ K}$$

Note that this is an expansion, and Mach number increases while pressure and temperature decrease, as expected.

***Discussion*** There are compressible flow calculators on the Internet that solve these implicit equations that arise in the analysis of compressible flow, along with both normal and oblique shock equations; e.g., see www.aoe.vt.edu/~devenpor/aoe3114/calc.html .

**12-95E**

**Solution** Air flowing at a specified supersonic Mach number is forced to undergo a compression turn (an oblique shock)., The Mach number, pressure, and temperature downstream of the oblique shock are to be determined.

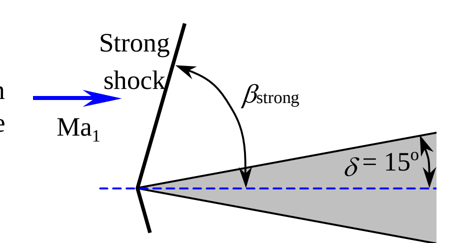



***Assumptions*** **1** The flow is steady. **2** The boundary layer on the wedge is very thin. **3** Air is an ideal gas with constant specific heats.

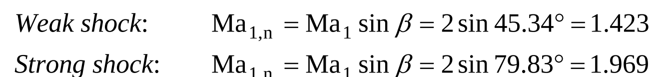
***Properties*** The specific heat ratio of air is $k = 1.4$.

***Analysis*** On the basis of Assumption #2, we take the deflection angle as equal to the wedge half-angle, i.e., $\theta \approx \delta = 15^o$. Then the two values of oblique shock angle $\beta$ are determined from

$$\tan\theta = \frac{2(\text{Ma}_1^2 \sin^2\beta - 1)/\tan\beta}{\text{Ma}_1^2 (k + \cos 2\beta) + 2} \quad \rightarrow \quad \tan 15° = \frac{2(2^2 \sin^2\beta - 1)/\tan\beta}{2^2 (1.4 + \cos 2\beta) + 2}$$




which is implicit in $\beta$. Therefore, we solve it by an iterative approach or with an equation solver such as EES. It gives $\beta_{weak} = 45.34^o$ and $\beta_{strong} = 79.83^o$. Then the upstream "normal" Mach number $Ma_{1,n}$ becomes



*Weak shock*: $\text{Ma}_{1,\text{n}} = \text{Ma}_1 \sin\beta = 2\sin 45.34° = 1.423$

*Strong shock*: $\text{Ma}_{1,\text{n}} = \text{Ma}_1 \sin\beta = 2\sin 79.83° = 1.969$



Also, the downstream normal Mach numbers $Ma_{2,n}$ become

*Weak shock*: $$\text{Ma}_{2,\text{n}} = \sqrt{\frac{(k-1)\text{Ma}_{1,\text{n}}^2 + 2}{2k\text{Ma}_{1,\text{n}}^2 - k + 1}} = \sqrt{\frac{(1.4-1)(1.423)^2 + 2}{2(1.4)(1.423)^2 - 1.4 + 1}} = 0.7304$$

*Strong shock*: $$\text{Ma}_{2,\text{n}} = \sqrt{\frac{(k-1)\text{Ma}_{1,\text{n}}^2 + 2}{2k\text{Ma}_{1,\text{n}}^2 - k + 1}} = \sqrt{\frac{(1.4-1)(1.969)^2 + 2}{2(1.4)(1.969)^2 - 1.4 + 1}} = 0.5828$$

The downstream pressure and temperature for each case are determined to be

*Weak shock*: $$P_2 = P_1 \frac{2k\text{Ma}_{1,\text{n}}^2 - k + 1}{k+1} = (6 \text{ psia}) \frac{2(1.4)(1.423)^2 - 1.4 + 1}{1.4 + 1} = \mathbf{13.2 \text{ psia}}$$

$$T_2 = T_1 \frac{P_2}{P_1}\frac{\rho_1}{\rho_2} = T_1 \frac{P_2}{P_1} \frac{2 + (k-1)\text{Ma}_{1,\text{n}}^2}{(k+1)\text{Ma}_{1,\text{n}}^2} = (480 \text{ R}) \frac{13.2 \text{ psia}}{6 \text{ psia}} \frac{2 + (1.4-1)(1.423)^2}{(1.4+1)(1.423)^2} = \mathbf{609 \text{ R}}$$

*Strong shock*: $$P_2 = P_1 \frac{2k\text{Ma}_{1,\text{n}}^2 - k + 1}{k+1} = (6 \text{ psia}) \frac{2(1.4)(1.969)^2 - 1.4 + 1}{1.4 + 1} = \mathbf{26.1 \text{ psia}}$$

$$T_2 = T_1 \frac{P_2}{P_1}\frac{\rho_1}{\rho_2} = T_1 \frac{P_2}{P_1} \frac{2 + (k-1)\text{Ma}_{1,\text{n}}^2}{(k+1)\text{Ma}_{1,\text{n}}^2} = (480 \text{ R}) \frac{26.1 \text{ psia}}{6 \text{ psia}} \frac{2 + (1.4-1)(1.969)^2}{(1.4+1)(1.969)^2} = \mathbf{798 \text{ R}}$$

The downstream Mach number is determined to be

*Weak shock*: $$\text{Ma}_2 = \frac{\text{Ma}_{2,\text{n}}}{\sin(\beta - \theta)} = \frac{0.7304}{\sin(45.34° - 15°)} = \mathbf{1.45}$$

*Strong shock*: $$\text{Ma}_2 = \frac{\text{Ma}_{2,\text{n}}}{\sin(\beta - \theta)} = \frac{0.5828}{\sin(79.83° - 15°)} = \mathbf{0.644}$$

***Discussion*** Note that the change in Mach number, pressure, temperature across the *strong shock* are much greater than the changes across the *weak shock*, as expected. For both the weak and strong oblique shock cases, $Ma_{1,n}$ is supersonic and $Ma_{2,n}$ is subsonic. However, $Ma_2$ is *supersonic* across the weak oblique shock, but *subsonic* across the strong oblique shock.

## Duct Flow with Heat Transfer and Negligible Friction (Rayleigh Flow)

### 12-96C

**Solution** We are to discuss the characteristic aspect of Rayleigh flow, and its main assumptions.

***Analysis*** The characteristic aspect of Rayleigh flow is **its involvement of heat transfer**. The main assumptions associated with Rayleigh flow are: the flow is **steady**, **one-dimensional**, and **frictionless** through a constant-area duct, and the fluid is an **ideal gas with constant specific heats**.

***Discussion*** Of course, there is no such thing as frictionless flow. It is better to say that frictional effects are negligible compared to the heating effects.

### 12-97C

**Solution** We are to discuss what the points on a *T-s* diagram of Rayleigh flow represent.

***Analysis*** The points on the Rayleigh line represent the **states that satisfy the conservation of mass, momentum, and energy equations as well as the property relations for a given state**. Therefore, for a given inlet state, the fluid cannot exist at any downstream state outside the Rayleigh line on a *T-s* diagram.

***Discussion*** The *T-s* diagram is quite useful, since any downstream state must lie on the Rayleigh line.

### 12-98C

**Solution** We are to discuss the effect of heat gain and heat loss on entropy during Rayleigh flow.

***Analysis*** In Rayleigh flow, **the effect of heat gain is to increase the entropy** of the fluid, and **the effect of heat loss is to decrease the entropy**.

***Discussion*** You should recall from thermodynamics that the entropy of a system can be lowered by removing heat.

### 12-99C

**Solution** We are to discuss how temperature and stagnation temperature change in subsonic Rayleigh flow.

***Analysis*** In Rayleigh flow, the **stagnation temperature $T_0$ always increases with heat transfer to the fluid**, but the temperature $T$ decreases with heat transfer in the Mach number range of $0.845 < \text{Ma} < 1$ for air. Therefore, the **temperature in this case will decrease**.

***Discussion*** This at first seems counterintuitive, but if heat were *not* added, the temperature would drop even *more* if the air were accelerated isentropically from Ma = 0.92 to 0.95.

### 12-100C

**Solution** We are to discuss the effect of heating on the flow velocity in subsonic Rayleigh flow.

***Analysis*** Heating the fluid **increases the flow velocity in subsonic Rayleigh flow**, but **decreases the flow velocity in supersonic Rayleigh flow**.

***Discussion*** These results are not necessarily intuitive, but must be true in order to satisfy the conservation laws.

### 12-101C

**Solution** We are to examine the Mach number at the end of a choked duct in Rayleigh flow when more heat is added.

***Analysis*** The flow is choked, and thus the flow at the duct exit **remains sonic**.

***Discussion*** There is no mechanism for the flow to become supersonic in this case.

**12-102**

**Solution** Fuel is burned in a tubular combustion chamber with compressed air. For a specified exit Mach number, the exit temperature and the rate of fuel consumption are to be determined.

***Assumptions*** **1** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid. **2** Combustion is complete, and it is treated as a heat addition process, with no change in the chemical composition of flow. **3** The increase in mass flow rate due to fuel injection is disregarded.

***Properties*** We take the properties of air to be $k = 1.4$, $c_p = 1.005$ kJ/kg·K, and $R = 0.287$ kJ/kg·K.

***Analysis*** The inlet density and mass flow rate of air are

$$\rho_1 = \frac{P_1}{RT_1} = \frac{400 \text{ kPa}}{(0.287 \text{ kJ/kgK})(500 \text{ K})} = 2.787 \text{ kg/m}^3$$

$$\dot{m}_{air} = \rho_1 A_{c1} V_1 = (2.787 \text{ kg/m}^3)[\pi(0.12 \text{ m})^2/4](70 \text{ m/s}) = 2.207 \text{ kg/s}$$

$\dot{Q}$

$P_1 = 400$ kPa
$T_1 = 500$ K
$V_1 = 70$ m/s
COMBUSTOR TUBE
$T_2$, $V_2$

The stagnation temperature and Mach number at the inlet are

$$T_{01} = T_1 + \frac{V_1^2}{2c_p} = 500 \text{ K} + \frac{(70 \text{ m/s})^2}{2 \times 1.005 \text{ kJ/kg}\cdot\text{K}}\left(\frac{1 \text{ kJ/kg}}{1000 \text{ m}^2/\text{s}^2}\right) = 502.4 \text{ K}$$

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287 \text{ kJ/kg}\cdot\text{K})(500 \text{ K})\left(\frac{1000 \text{ m}^2/\text{s}^2}{1 \text{ kJ/kg}}\right)} = 448.2 \text{ m/s}$$

$$\text{Ma}_1 = \frac{V_1}{c_1} = \frac{70 \text{ m/s}}{448.2 \text{ m/s}} = 0.1562$$

The Rayleigh flow functions corresponding to the inlet and exit Mach numbers are (Table A-15):

$\text{Ma}_1 = 0.1562$: $T_1/T^* = 0.1314$, $T_{01}/T^* = 0.1100$, $V_1/V^* = 0.0566$

$\text{Ma}_2 = 0.8$: $T_2/T^* = 1.0255$, $T_{02}/T^* = 0.9639$, $V_2/V^* = 0.8101$

The exit temperature, stagnation temperature, and velocity are determined to be

$$\frac{T_2}{T_1} = \frac{T_2/T^*}{T_1/T^*} = \frac{1.0255}{0.1314} = 7.804 \quad \rightarrow \quad T_2 = 7.804T_1 = 7.804(500 \text{ K}) = 3903 \text{ K} \cong \mathbf{3900\ K}$$

$$\frac{T_{02}}{T_{01}} = \frac{T_{02}/T^*}{T_{01}/T^*} = \frac{0.9639}{0.1100} = 8.763 \quad \rightarrow \quad T_{02} = 8.763T_{01} = 8.763(502.4 \text{ K}) = 4403 \text{ K}$$

$$\frac{V_2}{V_1} = \frac{V_2/V^*}{V_1/V^*} = \frac{0.8101}{0.0566} = 14.31 \quad \rightarrow \quad V_2 = 14.31V_1 = 14.31(70 \text{ m/s}) = 1002 \text{ m/s}$$

Then the mass flow rate of the fuel is determined to be

$$q = c_p(T_{02} - T_{01}) = (1.005 \text{ kJ/kg}\cdot\text{K})(4403 - 502.4) \text{ K} = 3920 \text{ kJ/kg}$$

$$\dot{Q} = \dot{m}_{air} q = (2.207 \text{ kg/s})(3920 \text{ kJ/kg}) = 8650 \text{ kW}$$

$$\dot{m}_{fuel} = \frac{\dot{Q}}{HV} = \frac{8650 \text{ kJ/s}}{39{,}000 \text{ kJ/kg}} = \mathbf{0.222\ kg/s}$$

***Discussion*** Note that both the temperature and velocity increase during this subsonic Rayleigh flow with heating, as expected. This problem can also be solved using appropriate relations instead of tabulated values, which can likewise be coded for convenient computer solutions.

**12-103**

**Solution** Fuel is burned in a rectangular duct with compressed air. For specified heat transfer, the exit temperature and Mach number are to be determined.

***Assumptions*** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid.

***Properties*** We take the properties of air to be $k = 1.4$, $c_p = 1.005$ kJ/kg·K, and $R = 0.287$ kJ/kg·K.

***Analysis*** The stagnation temperature and Mach number at the inlet are

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\ \text{kJ/kg}\cdot\text{K})(300\ \text{K})\left(\frac{1000\ \text{m}^2/\text{s}^2}{1\ \text{kJ/kg}}\right)} = 347.2\ \text{m/s}$$

$$V_1 = \text{Ma}_1 c_1 = 2(347.2\ \text{m/s}) = 694.4\ \text{m/s}$$

$$T_{01} = T_1 + \frac{V_1^2}{2c_p} = 300\ \text{K} + \frac{(694.4\ \text{m/s})^2}{2\times 1.005\ \text{kJ/kg}\cdot\text{K}}\left(\frac{1\ \text{kJ/kg}}{1000\ \text{m}^2/\text{s}^2}\right) = 539.9\ \text{K}$$

$q$ = 55 kJ/kg

$P_1$ = 420 kPa

$T_1$ = 300 K

$\text{Ma}_1 = 2$

$T_2$, $\text{Ma}_2$

The exit stagnation temperature is, from the energy equation $q = c_p(T_{02} - T_{01})$,

$$T_{02} = T_{01} + \frac{q}{c_p} = 539.9\ \text{K} + \frac{55\ \text{kJ/kg}}{1.005\ \text{kJ/kg}\cdot\text{K}} = 594.6\ \text{K}$$

The maximum value of stagnation temperature $T_0^*$ occurs at Ma = 1, and its value can be determined from Table A-15 or from the appropriate relation. At $\text{Ma}_1 = 2$ we read $T_{01}/T_0^* = 0.7934$. Therefore,

$$T_0^* = \frac{T_{01}}{0.7934} = \frac{539.9\ \text{K}}{0.7934} = 680.5\ \text{K}$$

The stagnation temperature ratio at the exit and the Mach number corresponding to it are, from Table A-15,

$$\frac{T_{02}}{T_0^*} = \frac{594.6\ \text{K}}{680.5\ \text{K}} = 0.8738 \quad \rightarrow \quad \text{Ma}_2 = 1.642 \cong \mathbf{1.64}$$

Also,

$\text{Ma}_1 = 2 \quad \rightarrow \quad T_1/T^* = 0.5289$

$\text{Ma}_2 = 1.642 \quad \rightarrow \quad T_2/T^* = 0.6812$

Then the exit temperature becomes

$$\frac{T_2}{T_1} = \frac{T_2/T^*}{T_1/T^*} = \frac{0.6812}{0.5289} = 1.288 \quad \rightarrow \quad T_2 = 1.288T_1 = 1.288(300\ \text{K}) = \mathbf{386\ K}$$

***Discussion*** Note that the temperature increases during this supersonic Rayleigh flow with heating. This problem can also be solved using appropriate relations instead of tabulated values, which can likewise be coded for convenient computer solutions.

**12-104**

**Solution** Compressed air is cooled as it flows in a rectangular duct. For specified heat rejection, the exit temperature and Mach number are to be determined.

***Assumptions*** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K.

***Analysis*** The stagnation temperature and Mach number at the inlet are

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(300\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 347.2\text{ m/s}$$

$$V_1 = \text{Ma}_1 c_1 = 2(347.2\text{ m/s}) = 694.4\text{ m/s}$$

$$T_{01} = T_1 + \frac{V_1^2}{2c_p} = 300\text{ K} + \frac{(694.4\text{ m/s})^2}{2\times 1.005\text{ kJ/kg}\cdot\text{K}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 539.9\text{ K}$$

q = -55 kJ/kg
$P_1$ = 420 kPa
$T_1$ = 300 K
$T_2$, $\text{Ma}_2$
$\text{Ma}_1$ = 2

The exit stagnation temperature is, from the energy equation $q = c_p(T_{02} - T_{01})$,

$$T_{02} = T_{01} + \frac{q}{c_p} = 539.9\text{ K} + \frac{-55\text{ kJ/kg}}{1.005\text{ kJ/kg}\cdot\text{K}} = 485.2\text{ K}$$

The maximum value of stagnation temperature $T_0^*$ occurs at Ma = 1, and its value can be determined from Table A-15 or from the appropriate relation. At $\text{Ma}_1$ = 2 we read $T_{01}/T_0^*$ = 0.7934. Therefore,

$$T_0^* = \frac{T_{01}}{0.7934} = \frac{539.9\text{ K}}{0.7934} = 680.5\text{ K}$$

The stagnation temperature ratio at the exit and the Mach number corresponding to it are, from Table A-15,

$$\frac{T_{02}}{T_0^*} = \frac{485.2\text{ K}}{680.5\text{ K}} = 0.7130 \quad\rightarrow\quad \text{Ma}_2 = 2.479 \cong \mathbf{2.48}$$

Also,

$\text{Ma}_1 = 2 \rightarrow T_1/T^* = 0.5289$

$\text{Ma}_2 = 2.479 \rightarrow T_2/T^* = 0.3838$

Then the exit temperature becomes

$$\frac{T_2}{T_1} = \frac{T_2/T^*}{T_1/T^*} = \frac{0.3838}{0.5289} = 0.7257 \quad\rightarrow\quad T_2 = 0.7257T_1 = 0.7257(300\text{ K}) = \mathbf{218\text{ K}}$$

***Discussion*** Note that the temperature decreases and Mach number increases during this supersonic Rayleigh flow with cooling. This problem can also be solved using appropriate relations instead of tabulated values, which can likewise be coded for convenient computer solutions.

**12-105**

**Solution** Air is heated in a duct during subsonic flow until it is choked. For specified pressure and velocity at the exit, the temperature, pressure, and velocity at the inlet are to be determined.

***Assumptions*** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid.

***Properties*** We take the properties of air to be $k = 1.4$, $c_p = 1.005$ kJ/kg·K, and $R = 0.287$ kJ/kg·K.

***Analysis*** Noting that sonic conditions exist at the exit, the exit temperature is

$$c_2 = V_2/\mathrm{Ma}_2 = (620\ \mathrm{m/s})/1 = 620\ \mathrm{m/s}$$

$$c_2 = \sqrt{kRT_2} \ \rightarrow\ \sqrt{(1.4)(0.287\ \mathrm{kJ/kg\cdot K})T_2\left(\frac{1000\ \mathrm{m^2/s^2}}{1\ \mathrm{kJ/kg}}\right)} = 620\ \mathrm{m/s}$$

q = 60 kJ/kg
$P_1$
$T_1$
$\mathrm{Ma}_1$
$P_2$= 270 kPa
$V_2$= 620 m/s
$\mathrm{Ma}_2$= 1

It gives $T_2 = 956.7$ K. Then the exit stagnation temperature becomes

$$T_{02} = T_2 + \frac{V_2^2}{2c_p} = 956.7\ \mathrm{K} + \frac{(620\ \mathrm{m/s})^2}{2\times 1.005\ \mathrm{kJ/kg\cdot K}}\left(\frac{1\ \mathrm{kJ/kg}}{1000\ \mathrm{m^2/s^2}}\right) = 1148\ \mathrm{K}$$

The inlet stagnation temperature is, from the energy equation $q = c_p(T_{02} - T_{01})$,

$$T_{01} = T_{02} - \frac{q}{c_p} = 1148\ \mathrm{K} - \frac{60\ \mathrm{kJ/kg}}{1.005\ \mathrm{kJ/kg\cdot K}} = 1088\ \mathrm{K}$$

The maximum value of stagnation temperature $T_0^*$ occurs at Ma = 1, and its value in this case is $T_{02}$ since the flow is choked. Therefore, $T_0^* = T_{02} = 1148$ K. Then the stagnation temperature ratio at the inlet, and the Mach number corresponding to it are, from Table A-15,

$$\frac{T_{01}}{T_0^*} = \frac{1088\ \mathrm{K}}{1148\ \mathrm{K}} = 0.9478 \quad \rightarrow \quad \mathrm{Ma}_1 = 0.7649 \cong \mathbf{0.765}$$

The Rayleigh flow functions corresponding to the inlet and exit Mach numbers are (Table A-15):

$\mathrm{Ma}_1 = 0.7649$: $T_1/T^* = 1.017$, $P_1/P^* = 1.319$, $V_1/V^* = 0.7719$
$\mathrm{Ma}_2 = 1$: $T_2/T^* = 1$, $P_2/P^* = 1$, $V_2/V^* = 1$

Then the inlet temperature, pressure, and velocity are determined to be

$$\frac{T_2}{T_1} = \frac{T_2/T^*}{T_1/T^*} = \frac{1}{1.017} \quad \rightarrow \quad T_1 = 1.017T_2 = 1.017(956.7\ \mathrm{K}) = \mathbf{974\ K}$$

$$\frac{P_2}{P_1} = \frac{P_2/P^*}{P_1/P^*} = \frac{1}{1.319} \quad \rightarrow \quad P_1 = 1.319P_2 = 1.319(270\ \mathrm{kPa}) = \mathbf{356\ kPa}$$

$$\frac{V_2}{V_1} = \frac{V_2/V^*}{V_1/V^*} = \frac{1}{0.7719} \quad \rightarrow \quad V_1 = 0.7719V_2 = 0.7719(620\ \mathrm{m/s}) = \mathbf{479\ m/s}$$

***Discussion*** Note that the temperature and pressure decreases with heating during this subsonic Rayleigh flow while velocity increases. This problem can also be solved using appropriate relations instead of tabulated values, which can likewise be coded for convenient computer solutions.

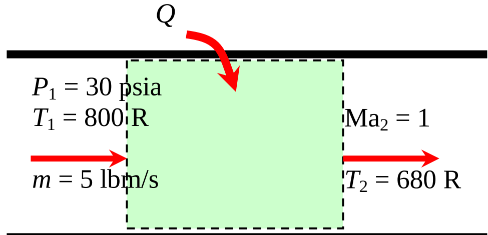
**12-106E**

**Solution** Air flowing with a subsonic velocity in a round duct is accelerated by heating until the flow is choked at the exit. The rate of heat transfer and the pressure drop are to be determined.

***Assumptions*** **1** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid. **2** The flow is choked at the duct exit. **3** Mass flow rate remains constant.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 0.2400 Btu/lbm·R, and $R$ = 0.06855 Btu/lbm·R = 0.3704 psia·ft$^3$/lbm·R.

***Analysis*** The inlet density and velocity of air are




$$\rho_1 = \frac{P_1}{RT_1} = \frac{30\text{ psia}}{(0.3704\text{ psia}\cdot\text{ft}^3/\text{lbm}\cdot\text{R})(800\text{ R})} = 0.1012\text{ lbm/ft}^3$$

$$V_1 = \frac{\dot{m}_{air}}{\rho_1 A_{c1}} = \frac{5\text{ lbm/s}}{(0.1012\text{ lbm/ft}^3)[\pi(4/12\text{ ft})^2/4]} = 565.9\text{ ft/s}$$

The stagnation temperature and Mach number at the inlet are

$$T_{01} = T_1 + \frac{V_1^2}{2c_p} = 800\text{ R} + \frac{(565.9\text{ ft/s})^2}{2\times 0.2400\text{ Btu/lbm}\cdot\text{R}}\left(\frac{1\text{ Btu/lbm}}{25{,}037\text{ ft}^2/\text{s}^2}\right) = 826.7\text{ R}$$

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.06855\text{ Btu/lbm}\cdot\text{R})(800\text{ R})\left(\frac{25{,}037\text{ ft}^2/\text{s}^2}{1\text{ Btu/lbm}}\right)} = 1386\text{ ft/s}$$

$$\text{Ma}_1 = \frac{V_1}{c_1} = \frac{565.9\text{ ft/s}}{1386\text{ ft/s}} = 0.4082$$

The Rayleigh flow functions corresponding to the inlet and exit Mach numbers are (Table A-15):

$\text{Ma}_1 = 0.4082$: $T_1/T^* = 0.6310$, $P_1/P^* = 1.946$, $T_{01}/T_0^* = 0.5434$

$\text{Ma}_2 = 1$: $T_2/T^* = 1$, $P_2/P^* = 1$, $T_{02}/T_0^* = 1$

Then the exit temperature, pressure, and stagnation temperature are determined to be

$$\frac{T_2}{T_1} = \frac{T_2/T^*}{T_1/T^*} = \frac{1}{0.6310} \quad \rightarrow \quad T_2 = T_1/0.6310 = (800\text{ R})/0.6310 = 1268\text{ R}$$

$$\frac{P_2}{P_1} = \frac{P_2/P^*}{P_1/P^*} = \frac{1}{1.946} \quad \rightarrow \quad P_2 = P_1/2.272 = (30\text{ psia})/1.946 = 15.4\text{ psia}$$

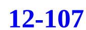
$$\frac{T_{02}}{T_{01}} = \frac{T_{02}/T^*}{T_{01}/T^*} = \frac{1}{0.5434} \quad \rightarrow \quad T_{02} = T_{01}/0.1743 = (826.7\text{ R})/0.5434 = 1521\text{ R}$$

Then the rate of heat transfer and the pressure drop become

$$\dot{Q} = \dot{m}_{\text{air}} c_p (T_{02} - T_{01}) = (5\text{ lbm/s})(0.2400\text{ Btu/lbm}\cdot\text{R})(1521 - 826.7)\text{ R} = \mathbf{834\ Btu/s}$$

$$\Delta P = P_1 - P_2 = 30 - 15.4 = \mathbf{14.6\ psia}$$

***Discussion*** Note that the entropy of air increases during this heating process, as expected.



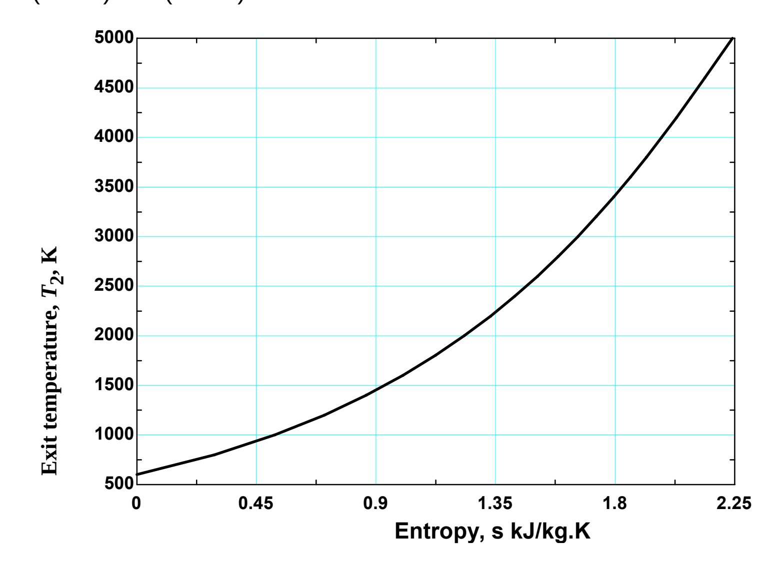
**12-107** 

**Solution** Air flowing with a subsonic velocity in a duct. The variation of entropy with temperature is to be investigated as the exit temperature varies from 600 K to 5000 K in increments of 200 K. The results are to be tabulated and plotted.

***Analysis*** We solve this problem using EES making use of Rayleigh functions. The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
k=1.4
cp=1.005
R=0.287
P1=350
T1=600
V1=70
C1=sqrt(k*R*T1*1000)
Ma1=V1/C1
T01=T1*(1+0.5*(k-1)*Ma1^2)
P01=P1*(1+0.5*(k-1)*Ma1^2)^(k/(k-1))
F1=1+0.5*(k-1)*Ma1^2
T01Ts=2*(k+1)*Ma1^2*F1/(1+k*Ma1^2)^2
P01Ps=((1+k)/(1+k*Ma1^2))*(2*F1/(k+1))^(k/(k-1))
T1Ts=(Ma1*((1+k)/(1+k*Ma1^2)))^2
P1Ps=(1+k)/(1+k*Ma1^2)
V1Vs=Ma1^2*(1+k)/(1+k*Ma1^2)
F2=1+0.5*(k-1)*Ma2^2
T02Ts=2*(k+1)*Ma2^2*F2/(1+k*Ma2^2)^2
P02Ps=((1+k)/(1+k*Ma2^2))*(2*F2/(k+1))^(k/(k-1))
T2Ts=(Ma2*((1+k)/(1+k*Ma2^2)))^2
P2Ps=(1+k)/(1+k*Ma2^2)
V2Vs=Ma2^2*(1+k)/(1+k*Ma2^2)
T02=T02Ts/T01Ts*T01
P02=P02Ps/P01Ps*P01
T2=T2Ts/T1Ts*T1
P2=P2Ps/P1Ps*P1
V2=V2Vs/V1Vs*V1
Delta_s=cp*ln(T2/T1)-R*ln(P2/P1)
```

| Exit temperature $T_2$, K | Exit Mach number, $Ma_2$ | Exit entropy relative to inlet, $s_2$, kJ/kg·K |
|---|---|---|
| 600 | 0.143 | 0.000 |
| 800 | 0.166 | 0.292 |
| 1000 | 0.188 | 0.519 |
| 1200 | 0.208 | 0.705 |
| 1400 | 0.227 | 0.863 |
| 1600 | 0.245 | 1.001 |
| 1800 | 0.263 | 1.123 |
| 2000 | 0.281 | 1.232 |
| 2200 | 0.299 | 1.331 |
| 2400 | 0.316 | 1.423 |
| 2600 | 0.333 | 1.507 |
| 2800 | 0.351 | 1.586 |
| 3000 | 0.369 | 1.660 |
| 3200 | 0.387 | 1.729 |
| 3400 | 0.406 | 1.795 |
| 3600 | 0.426 | 1.858 |
| 3800 | 0.446 | 1.918 |
| 4000 | 0.467 | 1.975 |
| 4200 | 0.490 | 2.031 |
| 4400 | 0.515 | 2.085 |
| 4600 | 0.541 | 2.138 |
| 4800 | 0.571 | 2.190 |
| 5000 | 0.606 | 2.242 |



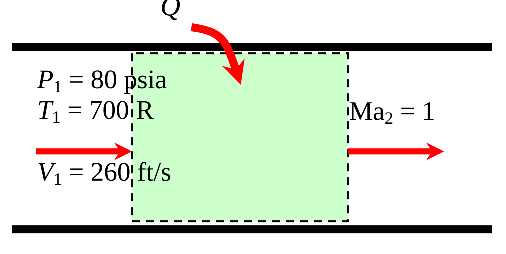

***Discussion*** Note that the entropy of air increases during this heating process, as expected.

**12-108E**

**Solution** Air flowing with a subsonic velocity in a square duct is accelerated by heating until the flow is choked at the exit. The rate of heat transfer and the entropy change are to be determined.

***Assumptions*** **1** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid. **2** The flow is choked at the duct exit. **3** Mass flow rate remains constant.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 0.2400 Btu/lbm·R, and $R$ = 0.06855 Btu/lbm·R = 0.3704 psia·ft³/lbm·R.




***Analysis*** The inlet density and mass flow rate of air are

$$\rho_1 = \frac{P_1}{RT_1} = \frac{80 \text{ psia}}{(0.3704 \text{ psia}\cdot\text{ft}^3/\text{lbm}\cdot\text{R})(700 \text{ R})} = 0.3085 \text{ lbm/ft}^3$$

$$\dot{m}_{air} = \rho_1 A_{c1} V_1 = (0.3085 \text{ lbm/ft}^3)(4\times 4/144 \text{ ft}^2)(260 \text{ ft/s}) = 8.914 \text{ lbm/s}$$

The stagnation temperature and Mach number at the inlet are

$$T_{01} = T_1 + \frac{V_1^2}{2c_p} = 700 \text{ R} + \frac{(260 \text{ ft/s})^2}{2\times 0.2400 \text{ Btu/lbm}\cdot\text{R}}\left(\frac{1 \text{ Btu/lbm}}{25{,}037 \text{ ft}^2/\text{s}^2}\right) = 705.6 \text{ R}$$

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.06855 \text{ Btu/lbm}\cdot\text{R})(700 \text{ R})\left(\frac{25{,}037 \text{ ft}^2/\text{s}^2}{1 \text{ Btu/lbm}}\right)} = 1297 \text{ ft/s}$$

$$\text{Ma}_1 = \frac{V_1}{c_1} = \frac{260 \text{ ft/s}}{1297 \text{ ft/s}} = 0.2005$$

The Rayleigh flow functions corresponding to the inlet and exit Mach numbers are (Table A-15):

$\text{Ma}_1 = 0.2005$: $T_1/T^* = 0.2075$, $P_1/P^* = 2.272$, $T_{01}/T_0^* = 0.1743$
$\text{Ma}_2 = 1$: $T_2/T^* = 1$, $P_2/P^* = 1$, $T_{02}/T_0^* = 1$

Then the exit temperature, pressure, and stagnation temperature are determined to be

$$\frac{T_2}{T_1} = \frac{T_2/T^*}{T_1/T^*} = \frac{1}{0.2075} \quad\rightarrow\quad T_2 = T_1/0.2075 = (700 \text{ R})/0.2075 = 3374 \text{ R}$$

$$\frac{P_2}{P_1} = \frac{P_2/P^*}{P_1/P^*} = \frac{1}{2.272} \quad\rightarrow\quad P_2 = P_1/2.272 = (80 \text{ psia})/2.272 = 35.2 \text{ psia}$$

$$\frac{T_{02}}{T_{01}} = \frac{T_{02}/T^*}{T_{01}/T^*} = \frac{1}{0.1743} \quad\rightarrow\quad T_{02} = T_{01}/0.1743 = (705.6 \text{ R})/0.1743 = 4048 \text{ R}$$

Then the rate of heat transfer and entropy change become

$$\dot{Q} = \dot{m}_{\text{air}} c_p \left(T_{02} - T_{01}\right) = (8.914 \text{ lbm/s})(0.2400 \text{ Btu/lbm}\cdot\text{R})(4048 - 705.6) \text{ R} = 7151 \text{ Btu/s} \cong \mathbf{7150 \text{ Btu/s}}$$

$$\Delta s = c_p \ln\frac{T_2}{T_1} - R\ln\frac{P_2}{P_1} = (0.2400 \text{ Btu/lbm}\cdot\text{R})\ln\frac{3374 \text{ R}}{700 \text{ R}} - (0.06855 \text{ Btu/lbm}\cdot\text{R})\ln\frac{35.2 \text{ psia}}{80 \text{ psia}} = \mathbf{0.434 \text{ Btu/lbm}\cdot\text{R}}$$

***Discussion*** Note that the entropy of air increases during this heating process, as expected.

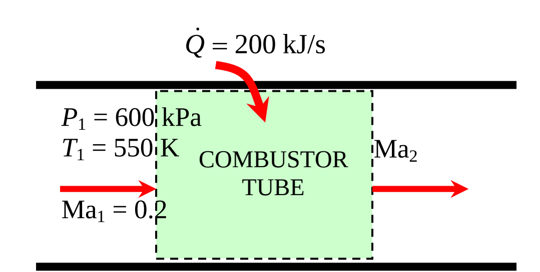
**12-109**

**Solution** Air enters the combustion chamber of a gas turbine at a subsonic velocity. For a specified rate of heat transfer, the Mach number at the exit and the loss in stagnation pressure to be determined.

***Assumptions*** **1** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid. **2** The cross-sectional area of the combustion chamber is constant. **3** The increase in mass flow rate due to fuel injection is disregarded.

***Properties*** We take the properties of air to be $k = 1.4$, $c_p = 1.005$ kJ/kg·K, and $R = 0.287$ kJ/kg·K.

***Analysis*** The inlet stagnation temperature and pressure are




$$T_{01} = T_1\left(1+\frac{k-1}{2}\text{Ma}_1^2\right) = (550\text{ K})\left(1+\frac{1.4\text{-}1}{2}0.2^2\right) = 554.4\text{ K}$$

$$P_{01} = P_1\left(1+\frac{k-1}{2}\text{ Ma}_1^2\right)^{k/(k-1)} = (600\text{ kPa})\left(1+\frac{1.4\text{-}1}{2}0.2^2\right)^{1.4/0.4}$$
$$= 617.0\text{ kPa}$$

The exit stagnation temperature is determined from

$$\dot{Q} = \dot{m}_{\text{air}} c_p (T_{02} - T_{01}) \quad \rightarrow \quad 200\text{ kJ/s} = (0.3\text{ kg/s})(1.005\text{ kJ/kg}\cdot\text{K})(T_{02} - 554.4)\text{ K}$$

It gives

$$T_{02} = 1218\text{ K.}$$

At $\text{Ma}_1 = 0.2$ we read from $T_{01}/T_0^* = 0.1736$ (Table A-15). Therefore,

$$T_0^* = \frac{T_{01}}{0.1736} = \frac{554.4\text{ K}}{0.1736} = 3193.5\text{ K}$$

Then the stagnation temperature ratio at the exit and the Mach number corresponding to it are (Table A-15)

$$\frac{T_{02}}{T_0^*} = \frac{1218\text{ K}}{3193.5\text{ K}} = 0.3814 \quad \rightarrow \quad \text{Ma}_2 = 0.3187 \cong \mathbf{0.319}$$

Also,

$$\text{Ma}_1 = 0.2 \quad \rightarrow \quad P_{01}/P_0^* = 1.2346$$
$$\text{Ma}_2 = 0.3187 \quad \rightarrow \quad P_{02}/P_0^* = 1.191$$

Then the stagnation pressure at the exit and the pressure drop become

$$\frac{P_{02}}{P_{01}} = \frac{P_{02}/P_0^*}{P_{01}/P_0^*} = \frac{1.191}{1.2346} = 0.9647 \quad \rightarrow \quad P_{02} = 0.9647P_{01} = 0.9647(617\text{ kPa}) = 595.2\text{ kPa}$$

and

$$\Delta P_0 = P_{01} - P_{02} = 617.0 - 595.2 = \mathbf{21.8\text{ kPa}}$$

***Discussion*** This problem can also be solved using appropriate relations instead of tabulated values, which can likewise be coded for convenient computer solutions.

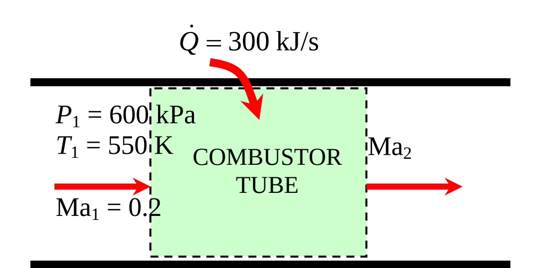
**12-110**

**Solution** Air enters the combustion chamber of a gas turbine at a subsonic velocity. For a specified rate of heat transfer, the Mach number at the exit and the loss in stagnation pressure to be determined.

***Assumptions*** **1** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid. **2** The cross-sectional area of the combustion chamber is constant. **3** The increase in mass flow rate due to fuel injection is disregarded.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K.

***Analysis*** The inlet stagnation temperature and pressure are




$$T_{01} = T_1\left(1+\frac{k-1}{2}\text{Ma}_1^2\right) = (550\text{ K})\left(1+\frac{1.4\text{-}1}{2}0.2^2\right) = 554.4\text{ K}$$

$$P_{01} = P_1\left(1+\frac{k-1}{2}\text{Ma}_1^2\right)^{k/(k-1)} = (600\text{ kPa})\left(1+\frac{1.4\text{-}1}{2}0.2^2\right)^{1.4/0.4}$$
$$= 617.0\text{ kPa}$$

The exit stagnation temperature is determined from

$$\dot{Q} = \dot{m}_{\text{air}} c_p (T_{02} - T_{01}) \quad \rightarrow \quad 300\text{ kJ/s} = (0.3\text{ kg/s})(1.005\text{ kJ/kg}\cdot\text{K})(T_{02} - 554.4)\text{ K}$$

It gives

$T_{02}$ = 1549 K.

At $Ma_1$ = 0.2 we read from $T_{01}/T_0^*$ = 0.1736 (Table A-15). Therefore,

$$T_0^* = \frac{T_{01}}{0.1736} = \frac{554.4\text{ K}}{0.1736} = 3193.5\text{ K}$$

Then the stagnation temperature ratio at the exit and the Mach number corresponding to it are (Table A-15)

$$\frac{T_{02}}{T_0^*} = \frac{1549\text{ K}}{3193.5\text{ K}} = 0.4850 \quad \rightarrow \quad \text{Ma}_2 = 0.3753 \cong \mathbf{0.375}$$

Also,

$Ma_1$ = 0.2 $\rightarrow$ $P_{01}/P_0^*$ = 1.2346
$Ma_2$ = 0.3753 $\rightarrow$ $P_{02}/P_0^*$ = 1.167

Then the stagnation pressure at the exit and the pressure drop become

$$\frac{P_{02}}{P_{01}} = \frac{P_{02}/P_0^*}{P_{01}/P_0^*} = \frac{1.167}{1.2346} = 0.9452 \quad \rightarrow \quad P_{02} = 0.9452 P_{01} = 0.9452(617\text{ kPa}) = 583.3\text{ kPa}$$

and

$$\Delta P_0 = P_{01} - P_{02} = 617.0 - 583.3 = \mathbf{33.7\text{ kPa}}$$

***Discussion*** This problem can also be solved using appropriate relations instead of tabulated values, which can likewise be coded for convenient computer solutions.

**12-111**

**Solution** Argon flowing at subsonic velocity in a constant-diameter duct is accelerated by heating. The highest rate of heat transfer without reducing the mass flow rate is to be determined.

***Assumptions*** **1** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid. **2** Mass flow rate remains constant.

***Properties*** We take the properties of argon to be $k$ = 1.667, $c_p$ = 0.5203 kJ/kg·K, and $R$ = 0.2081 kJ/kg·K.

$\dot{Q}$

$P_1$ = 320 kPa
$T_1$ = 400 K
$Ma_1$ = 0.2
$Ma_2$ = 1

***Analysis*** Heat transfer stops when the flow is choked, and thus $Ma_2 = V_2/c_2 = 1$. The inlet stagnation temperature is

$$T_{01} = T_1\left(1+\frac{k-1}{2}Ma_1^2\right) = (400\text{ K})\left(1+\frac{1.667\text{-}1}{2}0.2^2\right) = 405.3\text{ K}$$

The Rayleigh flow functions corresponding to the inlet and exit Mach numbers are

$T_{02}/T_0^* = 1$ (since $Ma_2 = 1$)

$$\frac{T_{01}}{T_0^*} = \frac{(k+1)Ma_1^2[2+(k-1)Ma_1^2]}{(1+kMa_1^2)^2} = \frac{(1.667+1)0.2^2[2+(1.667-1)0.2^2]}{(1+1.667\times 0.2^2)^2} = 0.1900$$ Therefore,

$$\frac{T_{02}}{T_{01}} = \frac{T_{02}/T_0^*}{T_{01}/T_0^*} = \frac{1}{0.1900} \quad \rightarrow \quad T_{02} = T_{01}/0.1900 = (405.3\text{ K})/0.1900 = 2133\text{ K}$$

Then the rate of heat transfer becomes

$$\dot{Q} = \dot{m}_{air}c_p(T_{02}-T_{01}) = (0.8\text{ kg/s})(0.5203\text{ kJ/kg}\cdot\text{K})(2133-400)\text{ K} = \mathbf{721\ kW}$$

***Discussion*** It can also be shown that $T_2$ = 1600 K, which is the highest thermodynamic temperature that can be attained under stated conditions. If more heat is transferred, the additional temperature rise will cause the mass flow rate to decrease. Also, in the solution of this problem, we cannot use the values of Table A-15 since they are based on $k$ = 1.4.

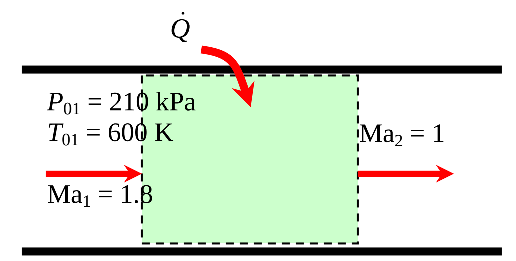
**12-112**

**Solution** Air flowing at a supersonic velocity in a duct is decelerated by heating. The highest temperature air can be heated by heat addition and the rate of heat transfer are to be determined.

***Assumptions*** **1**The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid. **2** Mass flow rate remains constant.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K.

***Analysis*** Heat transfer will stop when the flow is choked, and thus $Ma_2 = V_2/c_2 = 1$. Knowing stagnation properties, the static properties are determined to be




$$T_1 = T_{01}\left(1+\frac{k-1}{2}\text{Ma}_1^2\right)^{-1} = (600\text{ K})\left(1+\frac{1.4\text{-}1}{2}1.8^2\right)^{-1} = 364.1\text{ K}$$

$$P_1 = P_{01}\left(1+\frac{k-1}{2}\text{Ma}_1^2\right)^{-k/(k-1)} = (210\text{ kPa})\left(1+\frac{1.4\text{-}1}{2}1.8^2\right)^{-1.4/0.4} = 36.55\text{ kPa}$$

$$\rho_1 = \frac{P_1}{RT_1} = \frac{36.55\text{ kPa}}{(0.287\text{ kJ/kgK})(364.1\text{ K})} = 0.3498\text{ kg/m}^3$$

Then the inlet velocity and the mass flow rate become

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(364.1\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 382.5\text{ m/s}$$

$$V_1 = \text{Ma}_1 c_1 = 1.8(382.5\text{ m/s}) = 688.5\text{ m/s}$$

$$\dot{m}_{air} = \rho_1 A_{c1} V_1 = (0.3498\text{ kg/m}^3)[\pi(0.06\text{ m})^2/4](688.5\text{ m/s}) = 0.6809\text{ kg/s}$$

The Rayleigh flow functions corresponding to the inlet and exit Mach numbers are (Table A-15):

$Ma_1$ = 1.8: $T_1/T^*$ = 0.6089, $T_{01}/T_0^*$ = 0.8363

$Ma_2$ = 1: $T_2/T^*$ = 1, $T_{02}/T_0^*$ = 1

Then the exit temperature and stagnation temperature are determined to be

$$\frac{T_2}{T_1} = \frac{T_2/T^*}{T_1/T^*} = \frac{1}{0.6089} \quad\rightarrow\quad T_2 = T_1/0.6089 = (364.1\text{ K})/0.6089 = \mathbf{598\ K}$$

$$\frac{T_{02}}{T_{01}} = \frac{T_{02}/T_0^*}{T_{01}/T_0^*} = \frac{1}{0.8363} \quad\rightarrow\quad T_{02} = T_{01}/0.8363 = (600\text{ K})/0.8363 = 717.4\text{ K} \cong \mathbf{717\ K}$$

Finally, the rate of heat transfer is

$$\dot{Q} = \dot{m}_{\text{air}} c_p (T_{02} - T_{01}) = (0.6809\text{ kg/s})(1.005\text{ kJ/kg}\cdot\text{K})(717.4-600)\text{ K} = \mathbf{80.3\ kW}$$

***Discussion*** Note that this is the highest temperature that can be attained under stated conditions. If more heat is transferred, the additional temperature will cause the mass flow rate to decrease. Also, once the sonic conditions are reached, the thermodynamic temperature can be increased further by cooling the fluid and reducing the velocity (see the *T-s* diagram for Rayleigh flow).

## Adiabatic Duct Flow with Friction (Fanno Flow)

### 12-113C

**Solution** We are to discuss the characteristic aspect of Fanno flow and its main assumptions.

***Analysis*** The characteristic aspect of Fanno flow is its **consideration of friction**. The main assumptions associated with Fanno flow are: the flow is **steady**, **one-dimensional**, and **adiabatic** through a **constant-area duct**, and the fluid is an **ideal gas** with **constant specific heats**.

***Discussion*** Compared to Rayleigh flow, Fanno flow accounts for friction but neglects heat transfer effects, whereas Rayleigh flow accounts for heat transfer but neglects frictional effects.

### 12-114C

**Solution** We are to discuss the T-s diagram for Fanno flow.

***Analysis*** The points on the Fanno line on a *T-s* diagram represent the **states that satisfy the conservation of mass, momentum, and energy equations as well as the property relations for a given inlet state**. Therefore, for a given initial state, the fluid cannot exist at any downstream state outside the Fanno line on a *T-s* diagram.

***Discussion*** The *T-s* diagram is quite useful, since any downstream state must lie on the Fanno line.

### 12-115C

**Solution** We are to discuss the effect of friction on the entropy during Fanno flow.

***Analysis*** In Fanno flow, the effect of friction is **always to increase the entropy of the fluid**. Therefore Fanno flow always proceeds in the direction of increasing entropy.

***Discussion*** To do otherwise would violate the second law of thermodynamics.

### 12-116C

**Solution** We are to examine what happens when the Mach number of air increases in subsonic Fanno flow.

***Analysis*** During subsonic Fanno flow, the **stagnation temperature $T_0$ remains constant**, **stagnation pressure $P_0$ decreases**, and **entropy $s$ increases**.

***Discussion*** Friction leads to irreversible losses, which are felt as a loss of stagnation pressure and an increase of entropy. However, since the flow is adiabatic, the stagnation temperature does not change downstream.

### 12-117C

**Solution** We are to examine what happens when the Mach number of air decreases in supersonic Fanno flow.

***Analysis*** During supersonic Fanno flow, the **stagnation temperature $T_0$ remains constant**, **stagnation pressure $P_0$ decreases**, and **entropy $s$ increases**.

***Discussion*** Friction leads to irreversible losses, which are felt as a loss of stagnation pressure and an increase of entropy. However, since the flow is adiabatic, the stagnation temperature does not change downstream.

**12-118C**

**Solution** We are to discuss the effect of friction on velocity in Fanno flow.

***Analysis*** Friction **increases the flow velocity in subsonic Fanno flow**, but **decreases the flow velocity in supersonic flow**.

***Discussion*** These results may not be intuitive, but they come from following the Fanno line, which satisfies the conservation equations.

**12-119C**

**Solution** We are to discuss what happens to choked subsonic Fanno flow when the duct is extended.

***Analysis*** The flow is choked, and thus **the flow at the duct exit must remain sonic**. The **mass flow rate has to decrease** as a result of extending the duct length in order to compensate.

***Discussion*** Since there is no way for the flow to become supersonic (e.g., there is no throat), the upstream flow must adjust itself such that the flow at the exit plan remains sonic.

**12-120C**

**Solution** We are to discuss what happens to supersonic Fanno flow, initially sonic at the exit, when the duct is extended.

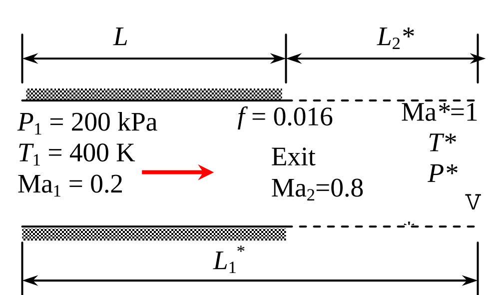
***Analysis*** **The flow at the duct exit remains sonic**. The **mass flow rate must remain constant** since upstream conditions are not affected by the added duct length.

***Discussion*** The mass flow rate is fixed by the upstream stagnation conditions and the size of the throat – therefore, the mass flow rate does not change by extending the duct. However, a *shock wave* appears in the duct when it is extended.

**12-121**

**Solution** Subsonic airflow in a constant cross-sectional area adiabatic duct is considered. For a specified exit Mach number, the duct length, temperature, pressure, and velocity at the duct exit are to be determined.

***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor remains constant along the duct.




***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K. The average friction factor is given to be $f$ = 0.016.

***Analysis*** The inlet velocity is

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\ \text{kJ/kg}\cdot\text{K})(400\ \text{K})\left(\frac{1000\ \text{m}^2/\text{s}^2}{1\ \text{kJ/kg}}\right)} = 400.9\ \text{m/s}$$

$$V_1 = \text{Ma}_1 c_1 = 0.2(400.9\ \text{m/s}) = 80.2\ \text{m/s}$$

The Fanno flow functions corresponding to the inlet and exit Mach numbers are, from Table A-16,

$\text{Ma}_1 = 0.2$: $(fL^*/D_h)_1 = 14.5333$ $T_1/T^* = 1.1905$, $P_1/P^* = 5.4554$, $V_1/V^* = 0.2182$

$\text{Ma}_2 = 0.8$: $(fL^*/D_h)_2 = 0.0723$ $T_2/T^* = 1.0638$, $P_2/P^* = 1.2893$, $V_2/V^* = 0.8251$

Then the temperature, pressure, and velocity at the duct exit are determined to be

$$\frac{T_2}{T_1} = \frac{T_2/T^*}{T_1/T^*} = \frac{1.0638}{1.1905} = 0.8936 \quad \rightarrow \quad T_2 = 0.8936T_1 = 0.8936(400\ \text{K}) = \mathbf{357\ K}$$

$$\frac{P_2}{P_1} = \frac{P_2/P^*}{P_1/P^*} = \frac{1.2893}{5.4554} = 0.2363 \quad \rightarrow \quad P_2 = 0.2363P_1 = 0.2363(200\ \text{kPa}) = \mathbf{47.3\ kPa}$$

$$\frac{V_2}{V_1} = \frac{V_2/V^*}{V_1/V^*} = \frac{0.8251}{0.2182} = 3.7814 \quad \rightarrow \quad V_2 = 3.7814V_1 = 3.7814(80.2\ \text{m/s}) = \mathbf{303\ m/s}$$

Finally, the actual duct length is determined to be

$$L = L_1^* - L_2^* = \left(\frac{fL_1^*}{D_h} - \frac{fL_2^*}{D_h}\right)\frac{D_h}{f} = (14.5333 - 0.0723)\frac{0.05\ \text{m}}{0.016} = \mathbf{45.2\ m}$$

***Discussion*** Note that it takes a duct length of 45.2 m for the Mach number to increase from 0.2 to 0.8. The Mach number rises at a much higher rate as sonic conditions are approached. The maximum (or sonic) duct lengths at the inlet and exit states in this case are $L_1^*$ = 45.4 m and $L_2^*$ = 0.2 m. Therefore, the flow would reach sonic conditions if a 0.2-m long section were added to the existing duct.

**12-122**

**Solution** Air enters a constant-area adiabatic duct of given length at a specified state. The exit Mach number, exit velocity, and the mass flow rate are to be determined.

***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor is constant along the duct.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K. The friction factor is given to be $f$ = 0.023.

L = 15 m
$L_2$*
$P_1$ = 300 kPa
$T_1$ = 500 K
$V_1$ =70 m/s
Exit
$Ma_2$
Ma*=1
T*
P*
V
$L_1^*$

***Analysis*** The first thing we need to know is whether the flow is choked at the exit or not. Therefore, we first determine the inlet Mach number and the corresponding value of the function $fL^*/D_h$ ,

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(500\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 448.2\text{ m/s}$$

$$\text{Ma}_1 = \frac{V_1}{c_1} = \frac{70\text{ m/s}}{448.2\text{ m/s}} = 0.1562$$

Corresponding to this Mach number we calculate (or read) from Table A-16), $(fL^*/D_h)_1$ = 25.540. Also, using the actual duct length $L$, we have

$$\frac{fL}{D_h} = \frac{(0.023)(15\text{ m})}{0.04\text{ m}} = 8.625 \quad < \quad 25.540$$

Therefore, flow is *not* choked and exit Mach number is less than 1. Noting that $L = L_1^* - L_2^*$, the function $fL^*/D_h$ at the exit state is calculated from

$$\left(\frac{fL^*}{D_h}\right)_2 = \left(\frac{fL^*}{D_h}\right)_1 - \frac{fL}{D_h} = 25.540 - 8.625 = 16.915$$

The Mach number corresponding to this value of $fL^*/D$ is obtained from Table A-16 to be

$\text{Ma}_2 =$ **0.187**

which is the Mach number at the duct exit. The mass flow rate of air is determined from the inlet conditions to be

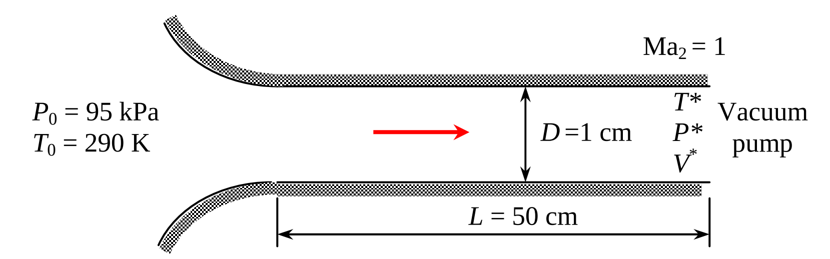
$$\rho_1 = \frac{P_1}{RT_1} = \frac{300\text{ kPa}}{(0.287\text{ kJ/kgK})(500\text{ K})}\left(\frac{1\text{ kJ}}{1\text{ kPa}\cdot\text{m}^3}\right) = 2.091\text{ kg/m}^3$$

$$\dot{m}_{air} = \rho_1 A_{c1} V_1 = (2.091\text{ kg/m}^3)[\pi(0.04\text{ m})^2/4](70\text{ m/s}) = \mathbf{0.184\ kg/s}$$

***Discussion*** It can be shown that $L_2^*$ = 29.4 m, indicating that it takes a duct length of 15 m for the Mach number to increase from 0.156 to 0.187, but only 29.4 m to increase from 0.187 to 1. Therefore, the Mach number rises at a much higher rate as sonic conditions are approached.

**12-123**

**Solution** Air enters a constant-area adiabatic duct at a specified state, and leaves at sonic state. The maximum duct length that will not cause the mass flow rate to be reduced is to be determined.




***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor is constant along the duct.

***Properties*** We take the properties of air to be $k = 1.4$, $c_p = 1.005$ kJ/kg·K, and $R = 0.287$ kJ/kg·K. The friction factor is given to be $f = 0.018$.

***Analysis*** The mass flow rate will be maximum when the flow is choked, and thus the exit Mach number is $Ma_2 = 1$. In that case we have

$$\frac{fL_1^*}{D} = \frac{fL_1}{D} = \frac{(0.018)(0.50\text{ m})}{0.01\text{ m}} = 0.9$$

The Mach number corresponding to this value of $fL^*/D$ at the tube inlet is obtained from Table A-16 to be $Ma_1 =$ **0.5225**. Noting that the flow in the nozzle section is isentropic, the thermodynamic temperature, pressure, and density at the tube inlet become

$$T_1 = T_{01}\left(1+\frac{k-1}{2}\text{Ma}_1^2\right)^{-1} = (290\text{ K})\left(1+\frac{1.4-1}{2}(0.5225)^2\right)^{-1} = 275.0\text{ K}$$

$$P_1 = P_{01}\left(1+\frac{k-1}{2}\text{Ma}_1^2\right)^{-k/(k-1)} = (95\text{ kPa})\left(1+\frac{1.4-1}{2}(0.5225)^2\right)^{-1.4/0.4} = 78.87\text{ kPa}$$

$$\rho_1 = \frac{P_1}{RT_1} = \frac{78.87\text{ kPa}}{(0.287\text{ kJ/kgK})(275.0\text{ K})} = 0.9993\text{ kg/m}^3$$

Then the inlet velocity and the mass flow rate become

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(275\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 332.4\text{ m/s}$$

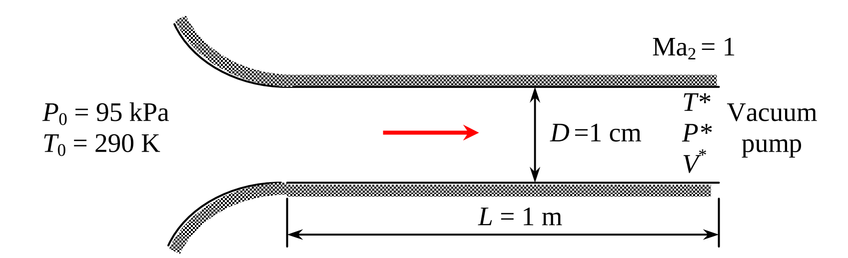
$$V_1 = \text{Ma}_1 c_1 = 0.5225(332.4\text{ m/s}) = 173.7\text{ m/s}$$

$$\dot{m}_{air} = \rho_1 A_{c1} V_1 = (0.9993\text{ kg/m}^3)[\pi(0.01\text{ m})^2/4](173.7\text{ m/s}) = \mathbf{0.0136\ kg/s}$$

***Discussion*** This is the maximum mass flow rate through the tube for the specified stagnation conditions at the inlet. The flow rate will remain at this level even if the vacuum pump drops the pressure even further.

**12-124**

**Solution** Air enters a constant-area adiabatic duct at a specified state, and leaves at sonic state. The maximum duct length that will not cause the mass flow rate to be reduced is to be determined.




***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor is constant along the duct.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K. The friction factor is given to be $f$ = 0.025.

***Analysis*** The mass flow rate will be maximum when the flow is choked, and thus the exit Mach number is $\text{Ma}_2$ = 1. In that case we have

$$\frac{fL_1^*}{D} = \frac{fL_1}{D} = \frac{(0.025)(1\,\text{m})}{0.01\,\text{m}} = 2.5$$

The Mach number corresponding to this value of $fL^*/D$ at the tube inlet is obtained from Table A-16 to be $\text{Ma}_1$ = 0.3899. Noting that the flow in the nozzle section is isentropic, the thermodynamic temperature, pressure, and density at the tube inlet become

$$T_1 = T_{01}\left(1+\frac{k-1}{2}\text{Ma}_1^2\right)^{-1} = (290\text{ K})\left(1+\frac{1.4-1}{2}(0.3899)^2\right)^{-1} = 281.4\text{ K}$$

$$P_1 = P_{01}\left(1+\frac{k-1}{2}\text{Ma}_1^2\right)^{-k/(k-1)} = (95\text{ kPa})\left(1+\frac{1.4-1}{2}(0.3899)^2\right)^{-1.4/0.4} = 85.54\text{ kPa}$$

$$\rho_1 = \frac{P_1}{RT_1} = \frac{85.54\text{ kPa}}{(0.287\text{ kJ/kgK})(281.4\text{ K})} = 1.059\text{ kg/m}^3$$

Then the inlet velocity and the mass flow rate become

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(281.4\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 336.3\text{ m/s}$$

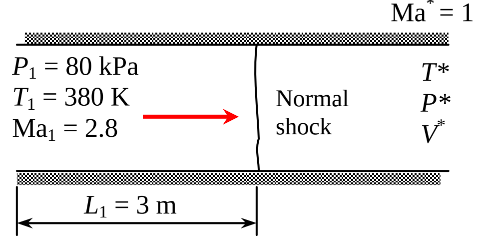
$$V_1 = \text{Ma}_1 c_1 = 0.3899(336.3\text{ m/s}) = 131.1\text{ m/s}$$

$$\dot{m}_{air} = \rho_1 A_{c1} V_1 = (1.059\text{ kg/m}^3)[\pi(0.01\text{ m})^2/4](131.1\text{ m/s}) = \mathbf{0.0109\ kg/s}$$

***Discussion*** This is the maximum mass flow rate through the tube for the specified stagnation conditions at the inlet. The flow rate will remain at this level even if the vacuum pump drops the pressure even further.

**12-125**

**Solution** Air enters a constant-area adiabatic duct at a specified state, and undergoes a normal shock at a specified location. The exit velocity, temperature, and pressure are to be determined.




***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor is constant along the duct.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K. The friction factor is given to be $f$ = 0.007.

***Analysis*** The Fanno flow functions corresponding to the inlet Mach number of 2.8 are, from Table A-16,

$\text{Ma}_1 = 2.8$: $(fL^*/D_h)_1 = 0.4898$ $T_1/T^* = 0.4673$, $P_1/P^* = 0.2441$

First we check to make sure that the flow everywhere upstream the shock is supersonic. The required duct length from the inlet $L_1^*$ for the flow to reach sonic conditions is

$$L_1^* = 0.4898\frac{D}{f} = 0.4898\frac{0.05\text{ m}}{0.007} = 3.50\text{ m}$$

which is greater than the actual length 3 m. Therefore, the flow is indeed supersonic when the normal shock occurs at the indicated location. Also, using the actual duct length $L_1$, we have $\frac{fL_1}{D_h} = \frac{(0.007)(3\text{ m})}{0.05\text{ m}} = 0.4200$. Noting that $L_1 = L_1^* - L_2^*$, the function $fL^*/D_h$ at the exit state and the corresponding Mach number are

$$\left(\frac{fL^*}{D_h}\right)_2 = \left(\frac{fL^*}{D_h}\right)_1 - \frac{fL_1}{D_h} = 0.4898 - 0.4200 = 0.0698 \quad\rightarrow\quad \text{Ma}_2 = 1.315$$

From Table A-16, at $\text{Ma}_2 = 1.315$: $T_2/T^* = 0.8918$ and $P_2/P^* = 0.7183$. Then the temperature, pressure, and velocity before the shock are determined to be

$$\frac{T_2}{T_1} = \frac{T_2/T^*}{T_1/T^*} = \frac{0.8918}{0.4673} = 1.9084 \quad\rightarrow\quad T_2 = 1.9084T_1 = 1.9084(380\text{ K}) = 725.2\text{ K}$$

$$\frac{P_2}{P_1} = \frac{P_2/P^*}{P_1/P^*} = \frac{0.7183}{0.2441} = 2.9426 \quad\rightarrow\quad P_2 = 2.9426P_1 = 2.9426(80\text{ kPa}) = 235.4\text{ kPa}$$

The normal shock functions corresponding to a Mach number of 1.315 are, from Table A-14,

$\text{Ma}_2 = 1.315$: $\text{Ma}_3 = 0.7786$, $T_3/T_2 = 1.2001$, $P_3/P_2 = 1.8495$

Then the temperature and pressure after the shock become

$$T_3 = 1.2001T_2 = 1.2001(725.2\text{ K}) = 870.3\text{ K} \quad\text{and}\quad P_3 = 1.8495P_2 = 1.8495(235.4\text{ kPa}) = 435.4\text{ kPa}$$

Sonic conditions exist at the duct exit, and the flow downstream the shock is still Fanno flow. From Table A-16,

$\text{Ma}_3 = 0.7786$: $T_3/T^* = 1.0702$, $P_3/P^* = 1.3286$
$\text{Ma}_4 = 1$: $T_4/T^* = 1$, $P_4/P^* = 1$

Then the temperature, pressure, and velocity at the duct exit are determined to be

$$\frac{T_4}{T_3} = \frac{T_4/T^*}{T_3/T^*} = \frac{1}{1.0702} \quad\rightarrow\quad T_4 = T_3/1.0702 = (870.3\text{ K})/1.0702 = \mathbf{813\text{ K}}$$

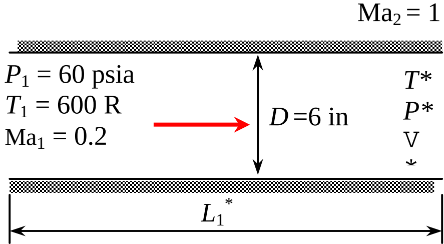
$$\frac{P_4}{P_3} = \frac{P_4/P^*}{P_3/P^*} = \frac{1}{1.3286} \quad\rightarrow\quad P_4 = P_3/1.3286 = (435.4\text{ kPa})/1.3286 = \mathbf{328\text{ kPa}}$$

$$V_4 = \text{Ma}_4 c_4 = (1)\sqrt{kRT_4} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(813\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = \mathbf{572\text{ m/s}}$$

***Discussion*** It can be shown that $L_3^*$ = 0.67 m, and thus the total length of this duct is 3.67 m. If the duct is extended, the normal shock will move further upstream, and eventually to the inlet of the duct.

## 12-126E

**Solution** Helium enters a constant-area adiabatic duct at a specified state, and leaves at sonic state. The maximum duct length that will not cause the mass flow rate to be reduced is to be determined.




***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor is constant along the duct.

***Properties*** We take the properties of helium to be $k$ = 1.667, $c_p$ = 1.2403 Btu/lbm·R, and $R$ = 0.4961 Btu/lbm·R. The friction factor is given to be $f$ = 0.025.

***Analysis*** The Fanno flow function *fL*/D* corresponding to the inlet Mach number of 0.2 is (Table A-16)

$$\frac{fL_1^*}{D} = 14.5333$$

Noting that * denotes sonic conditions, which exist at the exit state, the duct length is determined to be

$$L_1^* = 14.5333D/f = 14.5333(6/12 \text{ ft})/0.025 = \mathbf{291\ ft}$$

Thus, for the given friction factor, the duct length must be 291 ft for the Mach number to reach Ma = 1 at the duct exit.

***Discussion*** This problem can also be solved using equations instead of tabulated values for the Fanno functions.

---

## 12-127

**Solution** Subsonic airflow in a constant cross-sectional area adiabatic duct is considered. The duct length from the inlet where the inlet velocity doubles and the pressure drop in that section are to be determined.

$L$ $L_2^*$
$P_1$ = 200 kPa $f$ = 0.014 Ma*=1
$T_1$ = 500 K Exit $T^*$
$V_1$ =150 m/s $V_2$=2$V_1$ $P^*$
V
$L_1^*$

***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor remains constant along the duct.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K. The average friction factor is given to be $f$ = 0.014.

***Analysis*** The inlet Mach number is

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287 \text{ kJ/kg}\cdot\text{K})(500 \text{ K})\left(\frac{1000 \text{ m}^2/\text{s}^2}{1 \text{ kJ/kg}}\right)} = 448.2 \text{ m/s} \quad \rightarrow \quad \text{Ma}_1 = \frac{V_1}{c_1} = \frac{150 \text{ m/s}}{448.2 \text{ m/s}} = 0.3347$$

The Fanno flow functions corresponding to the inlet and exit Mach numbers are, from Table A-16,

$Ma_1$ = 0.3347: $(fL^*/D_h)_1$ = 3.924 $P_1/P^*$ = 3.2373, $V_1/V^*$ = 0.3626

Therefore, $V_1 = 0.3626V^*$. Then the Fanno function $V_2/V^*$ becomes $\dfrac{V_2}{V^*} = \dfrac{2V_1}{V^*} = \dfrac{2\times 0.3626V^*}{V^*} = 0.7252$ .

The corresponding Mach number and Fanno flow functions are, from Table A-16,

$Ma_2$ = 0.693, $(fL^*/D_h)_1$ = 0.2220, and $P_2/P^*$ = 1.5099.

Then the duct length where the velocity doubles, the exit pressure, and the pressure drop become

$$L = L_1^* - L_2^* = \left(\frac{fL_1^*}{D_h} - \frac{fL_2^*}{D_h}\right)\frac{D_h}{f} = (3.924 - 0.2220)\frac{0.20 \text{ m}}{0.014} = \mathbf{52.9\ m}$$

$$\frac{P_2}{P_1} = \frac{P_2/P^*}{P_1/P^*} = \frac{1.5099}{3.2373} = 0.4664 \qquad \rightarrow P_2 = 0.4664P_1 = 0.4664(200 \text{ kPa}) = 93.3 \text{ kPa}$$

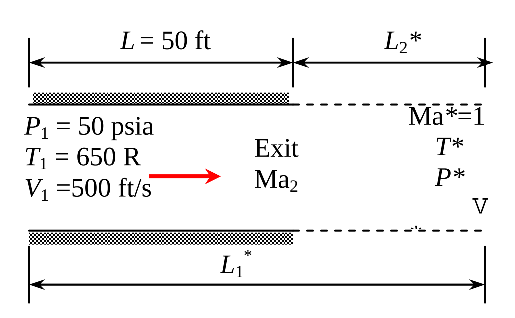
$$\Delta P = P_1 - P_2 = 200 - 93.3 = 106.7 \text{ kPa} \cong \mathbf{107\ \ kPa}$$

***Discussion*** Note that it takes a duct length of 52.9 m for the velocity to double, and the Mach number to increase from 0.3347 to 0.693. The maximum (or sonic) duct lengths at the inlet and exit states in this case are $L_1^*$ = 56.1 m and $L_2^*$ = 3.2 m. Therefore, the flow would reach sonic conditions if there is an additional 3.2 m of duct length.

---

**12-128E**

**Solution** Air enters a constant-area adiabatic duct of given length at a specified state. The velocity, temperature, and pressure at the duct exit are to be determined.




***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor is constant along the duct.

***Properties*** We take the properties of helium to be $k$ = 1.4, $c_p$ = 0.2400 Btu/lbm·R, and $R$ = 0.06855 Btu/lbm·R = 0.3704 psia·ft$^3$/lbm·R. The friction factor is given to be $f$ = 0.025.

***Analysis*** The first thing we need to know is whether the flow is choked at the exit or not. Therefore, we first determine the inlet Mach number and the corresponding value of the function $fL^*/D_h$ ,

$$T_1 = T_{01} - \frac{V_1^2}{2c_p} = 650\text{ R} - \frac{(500\text{ ft/s})^2}{2\times 0.2400\text{ Btu/lbm}\cdot\text{R}}\left(\frac{1\text{ Btu/lbm}}{25{,}037\text{ ft}^2/\text{s}^2}\right) = 629.2\text{ R}$$

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.06855\text{ Btu/lbm}\cdot\text{R})(629.2\text{ R})\left(\frac{25{,}037\text{ ft}^2/\text{s}^2}{1\text{ Btu/1bm}}\right)} = 1230\text{ ft/s}$$

$$\text{Ma}_1 = \frac{V_1}{c_1} = \frac{500\text{ m/s}}{1230\text{ ft/s}} = 0.4066$$

Corresponding to this Mach number we calculate (or read) from Table A-16), $(fL^*/D_h)_1$ = 2.1911. Also, using the actual duct length $L$, we have

$$\frac{fL}{D_h} = \frac{(0.02)(50\text{ ft})}{6/12\text{ ft}} = 2 \quad < \quad 2.1911$$

Therefore, the flow is *not* choked and exit Mach number is less than 1. Noting that $L = L_1^* - L_2^*$, the function $fL^*/D_h$ at the exit state is calculated from

$$\left(\frac{fL^*}{D_h}\right)_2 = \left(\frac{fL^*}{D_h}\right)_1 - \frac{fL}{D_h} = 2.1911 - 2 = 0.1911$$

The Mach number corresponding to this value of $fL^*/D$ is obtained from Table A-16 to be $\text{Ma}_2$ = 0.7091.

The Fanno flow functions corresponding to the inlet and exit Mach numbers are, from Table A-16,

$\text{Ma}_1 = 0.4066$: $T_1/T^* = 1.1616$, $P_1/P^* = 2.6504$, $V_1/V^* = 0.4383$

$\text{Ma}_2 = 0.7091$: $T_2/T^* = 1.0903$, $P_2/P^* = 1.4726$, $V_2/V^* = 0.7404$

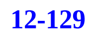
Then the temperature, pressure, and velocity at the duct exit are determined to be

$$\frac{T_2}{T_1} = \frac{T_2/T^*}{T_1/T^*} = \frac{1.0903}{1.1616} = 0.9386 \quad \rightarrow \quad T_2 = 0.9386T_1 = 0.9386(629.2\text{ R}) = \mathbf{591\,R}$$

$$\frac{P_2}{P_1} = \frac{P_2/P^*}{P_1/P^*} = \frac{1.4726}{2.6504} = 0.5556 \quad \rightarrow \quad P_2 = 0.5556P_1 = 0.5556(50\text{ psia}) = \mathbf{27.8\text{ psia}}$$

$$\frac{V_2}{V_1} = \frac{V_2/V^*}{V_1/V^*} = \frac{0.7404}{0.4383} = 1.6893 \quad \rightarrow \quad V_2 = 1.6893V_1 = 1.6893(500\text{ ft/s}) = \mathbf{845\text{ ft/s}}$$

***Discussion*** It can be shown that $L_2^*$ = 4.8 ft, indicating that it takes a duct length of 50 ft for the Mach number to increase from 0.4066 to 0.7091, but only 4.8 ft to increase from 0.7091 to 1. Therefore, the Mach number rises at a much higher rate as sonic conditions are approached.



**12-129** 

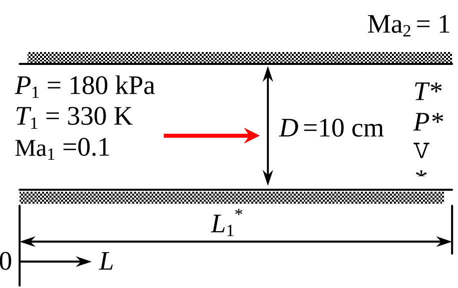
**Solution** Choked subsonic airflow in a constant cross-sectional area adiabatic duct is considered. The variation of duct length with Mach number is to be investigated, and the results are to be plotted.

***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor remains constant along the duct.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K. The average friction factor is given to be $f$ = 0.02.

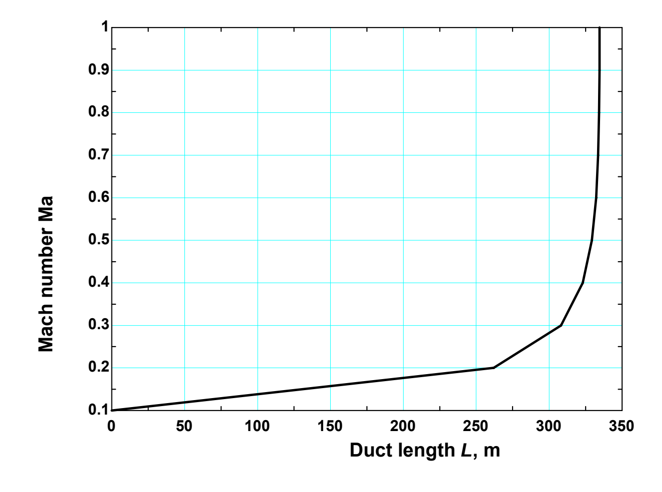
***Analysis*** The flow is choked, and thus $Ma_2$ =1. Corresponding to the inlet Mach number of $Ma_1$ = 0.1 we have, from Table A-16, $fL^*/D_h$ = 66.922, Therefore, the original duct length is

$$L_1^* = 66.922\frac{D}{f} = 66.922\frac{0.10\text{ m}}{0.02} = 335\text{ m}$$




Repeating the calculations for different $Ma_2$ as it varies from 0.1 to 1 results in the following table for the location on the duct from the inlet. The EES *Equations* window is printed below, along with the plotted results.

| Mach number, Ma | Duct length $L$, m |
|---|---|
| 0.10 | 0 |
| 0.20 | 262 |
| 0.30 | 308 |
| 0.40 | 323 |
| 0.50 | 329 |
| 0.60 | 332 |
| 0.70 | 334 |
| 0.80 | 334 |
| 0.90 | 335 |
| 1.00 | 335 |




**EES program:**

```
k=1.4
cp=1.005
R=0.287

P1=180
T1=330
Ma1=0.1
"Ma2=1"
f=0.02
D=0.1

C1=sqrt(k*R*T1*1000)
Ma1=V1/C1
T01=T02
T01=T1*(1+0.5*(k-1)*Ma1^2)
T02=T2*(1+0.5*(k-1)*Ma2^2)
P01=P1*(1+0.5*(k-1)*Ma1^2)^(k/(k-1))

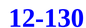
rho1=P1/(R*T1)
Ac=pi*D^2/4
mair=rho1*Ac*V1
```

```
P01Ps=((2+(k-1)*Ma1^2)/(k+1))^(0.5*(k+1)/(k-1))/Ma1
P1Ps=((k+1)/(2+(k-1)*Ma1^2))^0.5/Ma1
T1Ts=(k+1)/(2+(k-1)*Ma1^2)
R1Rs=((2+(k-1)*Ma1^2)/(k+1))^0.5/Ma1
V1Vs=1/R1Rs
fLs1=(1-Ma1^2)/(k*Ma1^2)+(k+1)/(2*k)*ln((k+1)*Ma1^2/(2+(k-1)*Ma1^2))
Ls1=fLs1*D/f

P02Ps=((2+(k-1)*Ma2^2)/(k+1))^(0.5*(k+1)/(k-1))/Ma2
P2Ps=((k+1)/(2+(k-1)*Ma2^2))^0.5/Ma2
T2Ts=(k+1)/(2+(k-1)*Ma2^2)
R2Rs=((2+(k-1)*Ma2^2)/(k+1))^0.5/Ma2
V2Vs=1/R2Rs
fLs2=(1-Ma2^2)/(k*Ma2^2)+(k+1)/(2*k)*ln((k+1)*Ma2^2/(2+(k-1)*Ma2^2))
Ls2=fLs2*D/f
L=Ls1-Ls2

P02=P02Ps/P01Ps*P01
P2=P2Ps/P1Ps*P1
V2=V2Vs/V1Vs*V1
```

***Discussion*** Note that the Mach number increases very mildly at the beginning, and then rapidly near the duct outlet. It takes 262 m of duct length for Mach number to increase from 0.1 to 0.2, but only 1 m to increase from 0.7 to 1.



**12-130** 

**Solution** The flow of argon gas in a constant cross-sectional area adiabatic duct is considered. The variation of entropy change with exit temperature is to be investigated, and the calculated results are to be plotted on a *T-s* diagram.

***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor remains constant along the duct.

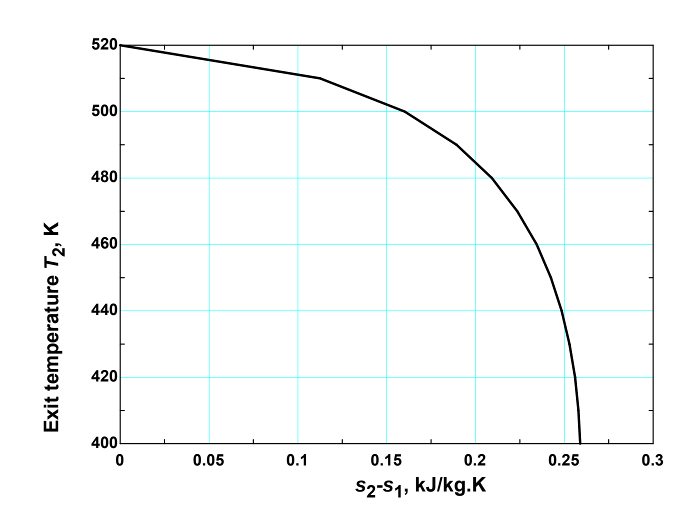
***Properties*** The properties of argon are given to be $k$ = 1.667, $c_p$ = 0.5203 kJ/kg·K, and $R$ = 0.2081 kJ/kg·K. The average friction factor is given to be $f$ = 0.005.

*L* (variable)

$P_1$ = 350 kPa
$T_1$ = 500 K
$V_1$ =70 m/s
$f$ = 0.005
Exit
$T_2$
Ma*=1
*T**
*P**

***Analysis*** Using EES, we determine the entropy change and tabulate and plot the results as follows:

| Exit temp. $T_2$, K | Mach umber $Ma_2$ | Entropy change $\Delta s$, kg/kg·K |
|---|---|---|
| 520 | 0.165 | 0.000 |
| 510 | 0.294 | 0.112 |
| 500 | 0.385 | 0.160 |
| 490 | 0.461 | 0.189 |
| 480 | 0.528 | 0.209 |
| 470 | 0.591 | 0.224 |
| 460 | 0.649 | 0.234 |
| 450 | 0.706 | 0.242 |
| 440 | 0.760 | 0.248 |
| 430 | 0.813 | 0.253 |
| 420 | 0.865 | 0.256 |
| 410 | 0.916 | 0.258 |
| 400 | 0.967 | 0.259 |




**EES Program:**

```
k=1.667
cp=0.5203
R=0.2081
P1=350
T1=520
V1=70
"T2=400"
f=0.005
D=0.08
C1=sqrt(k*R*T1*1000)
Ma1=V1/C1
T01=T02
T01=T1*(1+0.5*(k-1)*Ma1^2)
P01=P1*(1+0.5*(k-1)*Ma1^2)^(k/(k-1))
rho1=P1/(R*T1)
Ac=pi*D^2/4
mair=rho1*Ac*V1
P01Ps=((2+(k-1)*Ma1^2)/(k+1))^(0.5*(k+1)/(k-1))/Ma1
P1Ps=((k+1)/(2+(k-1)*Ma1^2))^0.5/Ma1
T1Ts=(k+1)/(2+(k-1)*Ma1^2)
R1Rs=((2+(k-1)*Ma1^2)/(k+1))^0.5/Ma1
V1Vs=1/R1Rs
fLs1=(1-Ma1^2)/(k*Ma1^2)+(k+1)/(2*k)*ln((k+1)*Ma1^2/(2+(k-1)*Ma1^2))
Ls1=fLs1*D/f

P02Ps=((2+(k-1)*Ma2^2)/(k+1))^(0.5*(k+1)/(k-1))/Ma2
```

```
P2Ps=((k+1)/(2+(k-1)*Ma2^2))^0.5/Ma2
T2Ts=(k+1)/(2+(k-1)*Ma2^2)
R2Rs=((2+(k-1)*Ma2^2)/(k+1))^0.5/Ma2
V2Vs=1/R2Rs
fLs2=(1-Ma2^2)/(k*Ma2^2)+(k+1)/(2*k)*ln((k+1)*Ma2^2/(2+(k-1)*Ma2^2))
Ls2=fLs2*D/f
L=Ls1-Ls2

P02=P02Ps/P01Ps*P01
P2=P2Ps/P1Ps*P1
T2=T2Ts/T1Ts*T1
V2=V2Vs/V1Vs*V1
Del_s=cp*ln(T2/T1)-R*ln(P2/P1)
```

***Discussion*** Note that entropy increases with increasing duct length and Mach number (and thus decreasing temperature). It reached a maximum value of 0.259 kJ/kg.K when the Mach number reaches $Ma_2 = 1$ and thus the flow is choked.

---

## Review Problems

### 12-131

**Solution** A leak develops in an automobile tire as a result of an accident. The initial mass flow rate of air through the leak is to be determined.

***Assumptions*** **1** Air is an ideal gas with constant specific heats. **2** Flow of air through the hole is isentropic.

***Properties*** For air at room temperature, the gas constant is $R$ = 0.287 kPa·m$^3$/kg·K, and the specific heat ratio is $k$ = 1.4.

***Analysis*** The absolute pressure in the tire is

$$P = P_{\text{gage}} + P_{\text{atm}} = 220 + 94 = 314\ \text{kPa}$$

The critical pressure is, from Table 12-2,

$$P^* = 0.5283 P_0 = (0.5283)(314\ \text{kPa}) = 166\ \text{kPa} > 94\ \text{kPa}$$

Therefore, the flow is choked, and the velocity at the exit of the hole is the sonic speed. Then the flow properties at the exit becomes

$$\rho_0 = \frac{P_0}{RT_0} = \frac{314\ \text{kPa}}{(0.287\ \text{kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(298\ \text{K})} = 3.671\ \text{kg/m}^3$$

$$\rho^* = \rho_0\left(\frac{2}{k+1}\right)^{1/(k-1)} = (3.671\ \text{kg/m}^3)\left(\frac{2}{1.4+1}\right)^{1/(1.4-1)} = 2.327\ \text{kg/m}^3$$

$$T^* = \frac{2}{k+1}T_0 = \frac{2}{1.4+1}(298\ \text{K}) = 248.3\ \text{K}$$

$$V = c = \sqrt{kRT^*} = \sqrt{(1.4)(0.287\ \text{kJ/kg}\cdot\text{K})\left(\frac{1000\ \text{m}^2/\text{s}^2}{1\ \text{kJ/kg}}\right)(248.3\ \text{K})} = 315.9\ \text{m/s}$$

Then the initial mass flow rate through the hole becomes

$$\dot{m} = \rho A V = (2.327\ \text{kg/m}^3)[\pi(0.004\ \text{m})^2/4](315.9\ \text{m/s}) = 0.00924\ \text{kg/s} = \mathbf{0.554\ kg/min}$$

***Discussion*** The mass flow rate will decrease with time as the pressure inside the tire drops.

---

**12-132**

**Solution** The thrust developed by the engine of a Boeing 777 is about 380 kN. The mass flow rate of gases through the nozzle is to be determined.

***Assumptions*** **1** Air is an ideal gas with constant specific heats. **2** Flow of combustion gases through the nozzle is isentropic. **3** Choked flow conditions exist at the nozzle exit. **4** The velocity of gases at the nozzle inlet is negligible.

***Properties*** The gas constant of air is $R$ = 0.287 kPa·m$^3$/kg·K, and it can also be used for combustion gases. The specific heat ratio of combustion gases is $k$ = 1.33.

***Analysis*** The velocity at the nozzle exit is the sonic speed, which is determined to be

$$V = c = \sqrt{kRT} = \sqrt{(1.33)(0.287\text{ kJ/kg}\cdot\text{K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)(295\text{ K})} = 335.6\text{ m/s}$$

Noting that thrust $F$ is related to velocity by $F = \dot{m}V$ , the mass flow rate of combustion gases is determined to be

$$\dot{m} = \frac{F}{V} = \frac{380{,}000\text{ N}}{335.6\text{ m/s}}\left(\frac{1\text{ kg.m/s}^2}{1\text{ N}}\right) = 1132\text{ kg/s} \cong \mathbf{1130\ \ kg/s}$$

***Discussion*** The combustion gases are mostly nitrogen (due to the 78% of $N_2$ in air), and thus they can be treated as air with a good degree of approximation.

---

**12-133**

**Solution** A stationary temperature probe is inserted into an air duct reads 85°C. The actual temperature of air is to be determined.

***Assumptions*** **1** Air is an ideal gas with constant specific heats at room temperature. **2** The stagnation process is isentropic.

***Properties*** The specific heat of air at room temperature is $c_p$ = 1.005 kJ/kg·K.

***Analysis*** The air that strikes the probe will be brought to a complete stop, and thus it will undergo a stagnation process. The thermometer will sense the temperature of this stagnated air, which is the stagnation temperature. The actual air temperature is determined from

$$T = T_0 - \frac{V^2}{2c_p} = 85°\text{C} - \frac{(250\text{ m/s})^2}{2\times 1.005\text{ kJ/kg}\cdot\text{K}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = \mathbf{53.9°C}$$

T

250 m/s

***Discussion*** Temperature rise due to stagnation is very significant in high-speed flows, and should always be considered when compressibility effects are not negligible.

---

**12-134**

**Solution** Nitrogen flows through a heat exchanger. The stagnation pressure and temperature of the nitrogen at the inlet and the exit states are to be determined.

***Assumptions*** **1** Nitrogen is an ideal gas with constant specific heats. **2** Flow of nitrogen through the heat exchanger is isentropic.

***Properties*** The properties of nitrogen are $c_p$ = 1.039 kJ/kg·K and $k$ = 1.4.

***Analysis*** The stagnation temperature and pressure of nitrogen at the inlet and the exit states are determined from

$$T_{01} = T_1 + \frac{V_1^2}{2c_p} = 10°\text{C} + \frac{(100\text{ m/s})^2}{2\times 1.039\text{ kJ/kg}\cdot°\text{C}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = \mathbf{14.8°C}$$

$$P_{01} = P_1\left(\frac{T_{01}}{T_1}\right)^{k/(k-1)} = (150\text{ kPa})\left(\frac{288.0\text{ K}}{283.2\text{ K}}\right)^{1.4/(1.4-1)} = \mathbf{159\ kPa}$$



From the energy balance relation $E_{in} - E_{out} = \Delta E_{system}$ with $w$ = 0

$$q_{in} = c_p(T_2 - T_1) + \frac{V_2^2 - V_1^2}{2} + \Delta pe^{\nearrow 0}$$

$$150\text{ kJ/kg} = (1.039\text{ kJ/kg}\cdot°\text{C})(T_2 - 10°\text{C}) + \frac{(200\text{ m/s})^2 - (100\text{ m/s})^2}{2}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right)$$

$$T_2 = 139.9°\text{C}$$

and

$$T_{02} = T_2 + \frac{V_2^2}{2c_p} = 139.9°\text{C} + \frac{(200\text{ m/s})^2}{2\times 1.039\text{ kJ/kg}\cdot°\text{C}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = \mathbf{159°C}$$

$$P_{02} = P_2\left(\frac{T_{02}}{T_2}\right)^{k/(k-1)} = (100\text{ kPa})\left(\frac{432.3\text{ K}}{413.1\text{ K}}\right)^{1.4/(1.4-1)} = \mathbf{117\ kPa}$$

***Discussion*** Note that the stagnation temperature and pressure can be very different than their thermodynamic counterparts when dealing with compressible flow.

**12-135**

**Solution** An expression for the speed of sound based on van der Waals equation of state is to be derived. Using this relation, the speed of sound in carbon dioxide is to be determined and compared to that obtained by ideal gas behavior.

***Properties*** The properties of $CO_2$ are $R$ = 0.1889 kJ/kg·K and $k$ = 1.279 at $T$ = 50°C = 323.2 K.

***Analysis*** Van der Waals equation of state can be expressed as $P = \dfrac{RT}{v-b} - \dfrac{a}{v^2}$ .

Differentiating, $$\left(\frac{\partial P}{\partial v}\right)_T = \frac{RT}{(v-b)^2} + \frac{2a}{v^3}$$

Noting that $\rho = 1/v \longrightarrow d\rho = -dv/v^2$ , the speed of sound relation becomes

$$c^2 = k\left(\frac{\partial P}{\partial r}\right)_T = v^2 k\left(\frac{\partial P}{\partial v}\right)_T$$

Substituting,

$$c^2 = \frac{v^2 kRT}{(v-b)^2} - \frac{2ak}{v}$$

Using the molar mass of $CO_2$ ($M$ = 44 kg/kmol), the constant a and b can be expressed per unit mass as

$$a = 0.1882\ \text{kPa}\cdot\text{m}^6/\text{kg}^2 \quad \text{and} \quad b = 9.70\times10^{-4}\ \text{m}^3/\text{kg}$$

The specific volume of $CO_2$ is determined to be

$$200\ \text{kPa} = \frac{(0.1889\ \text{kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(323.2\ \text{K})}{\text{v} - 0.000970\ \text{m}^3/\text{kg}} - \frac{2\times0.1882\ \text{kPa}\cdot\text{m}^6/\text{kg}^2}{v^2} \rightarrow v = 0.300\ \text{m}^3/\text{kg}$$

Substituting,

$$\text{c} = \left(\left(\frac{(0.300\ \text{m}^3/\text{kg})^2(1.279)(0.1889\ \text{kJ/kg}\cdot\text{K})(323.2\ \text{K})}{(0.300-0.000970\ \text{m}^3/\text{kg})^2} - \frac{2(0.1882\ \text{kPa}\cdot\text{m}^6/\text{kg}^3)(1.279)}{(0.300\ \text{m}^3/\text{kg})^2}\right)\left(\frac{1000\ \text{m}^2/\text{s}^2}{1\ \text{kPa}\cdot\text{m}^3/\text{kg}}\right)\right)^{1/2}$$

$$= \mathbf{271\ m/s}$$

If we treat $CO_2$ as an ideal gas, the speed of sound becomes

$$c = \sqrt{kRT} = \sqrt{(1.279)(0.1889\ \text{kJ/kg}\cdot\text{K})(323.2\ \text{K})\left(\frac{1000\ \text{m}^2/\text{s}^2}{1\ \text{kJ/kg}}\right)} = \mathbf{279\ m/s}$$

***Discussion*** Note that the ideal gas relation is the simplest equation of state, and it is very accurate for most gases encountered in practice. At high pressures and/or low temperatures, however, the gases deviate from ideal gas behavior, and it becomes necessary to use more complicated equations of state.

**12-136**

**Solution** The equivalent relation for the speed of sound is to be verified using thermodynamic relations.

***Analysis*** The two relations are $c^2 = \left(\frac{\partial P}{\partial \rho}\right)_s$ and $c^2 = k\left(\frac{\partial P}{\partial \rho}\right)_T$

From $r = 1/v \longrightarrow dr = -dv/v^2$. Thus, $c^2 = \left(\frac{\partial P}{\partial r}\right)_s = -v^2\left(\frac{\partial P}{\partial v}\right)_s = -v^2\left(\frac{\partial P}{\partial T}\frac{\partial T}{\partial v}\right)_s = -v^2\left(\frac{\partial P}{\partial T}\right)_s\left(\frac{\partial T}{\partial v}\right)_s$

From the cyclic rule,

$$(P,T,s):\left(\frac{\partial P}{\partial T}\right)_s\left(\frac{\partial T}{\partial s}\right)_P\left(\frac{\partial s}{\partial P}\right)_T = -1 \longrightarrow \left(\frac{\partial P}{\partial T}\right)_s = -\left(\frac{\partial s}{\partial T}\right)_P\left(\frac{\partial P}{\partial s}\right)_T$$

$$(T,v,s):\left(\frac{\partial T}{\partial v}\right)_s\left(\frac{\partial v}{\partial s}\right)_T\left(\frac{\partial s}{\partial T}\right)_v = -1 \longrightarrow \left(\frac{\partial T}{\partial v}\right)_s = -\left(\frac{\partial s}{\partial v}\right)_T\left(\frac{\partial T}{\partial s}\right)_v$$

Substituting,

$$c^2 = -v^2\left(\frac{\partial s}{\partial T}\right)_P\left(\frac{\partial P}{\partial s}\right)_T\left(\frac{\partial s}{\partial v}\right)_T\left(\frac{\partial T}{\partial s}\right)_v = -v^2\left(\frac{\partial s}{\partial T}\right)_P\left(\frac{\partial T}{\partial s}\right)_v\left(\frac{\partial P}{\partial v}\right)_T$$

Recall that $\frac{c_p}{T} = \left(\frac{\partial s}{\partial T}\right)_P$ and $\frac{c_v}{T} = \left(\frac{\partial s}{\partial T}\right)_v$. Substituting,

$$c^2 = -v^2\left(\frac{c_p}{T}\right)\left(\frac{T}{c_v}\right)\left(\frac{\partial P}{\partial v}\right)_T = -v^2 k\left(\frac{\partial P}{\partial v}\right)_T$$

Replacing $-dv/v^2$ by d$\rho$, we get $\boxed{c^2 = k\left(\frac{\partial P}{\partial \rho}\right)_T}$, which is the desired expression

***Discussion*** Note that the differential thermodynamic property relations are very useful in the derivation of other property relations in differential form.

---

**12-137**

**Solution** For ideal gases undergoing isentropic flows, expressions for $P/P^*$, $T/T^*$, and $\rho/\rho^*$ as functions of $k$ and Ma are to be obtained.

***Analysis*** Equations 12-18 and 12-21 are given to be $\frac{T_0}{T} = \frac{2+(k-1)\text{Ma}^2}{2}$ and $\frac{T^*}{T_0} = \frac{2}{k+1}$

Multiplying the two, $\left(\frac{T_0}{T}\frac{T^*}{T_0}\right) = \left(\frac{2+(k-1)\text{Ma}^2}{2}\right)\left(\frac{2}{k+1}\right)$

Simplifying and inverting, $\frac{T}{T^*} = \frac{k+1}{2+(k-1)\text{Ma}^2}$ (1)

From $\frac{P}{P^*} = \left(\frac{T}{T^*}\right)^{k/(k-1)} \longrightarrow \frac{P}{P^*} = \left(\frac{k+1}{2+(k-1)\text{Ma}^2}\right)^{k/(k-1)}$ (2)

From $\frac{\rho}{\rho^*} = \left(\frac{\rho}{\rho^*}\right)^{k/(k-1)} \longrightarrow \frac{\rho}{\rho^*} = \left(\frac{k+1}{2+(k-1)\text{Ma}^2}\right)^{k/(k-1)}$ (3)

***Discussion*** Note that some very useful relations can be obtained by very simple manipulations.

---

**12-138**

**Solution** It is to be verified that for the steady flow of ideal gases $dT_0/T = dA/A + (1\text{-Ma}^2)\, dV/V$. The effect of heating and area changes on the velocity of an ideal gas in steady flow for subsonic flow and supersonic flow are to be explained.

***Analysis*** We start with the relation $$\frac{V^2}{2} = c_p(T_0 - T) \qquad (1)$$

Differentiating, $$V\,dV = c_p(dT_0 - dT) \qquad (2)$$

We also have $$\frac{d\rho}{\rho} + \frac{dA}{A} + \frac{dV}{V} = 0 \qquad (3)$$

and $$\frac{dP}{\rho} + V\,dV = 0 \qquad (4)$$

Differentiating the ideal gas relation $P = \rho RT$, $$\frac{dP}{P} = \frac{d\rho}{\rho} + \frac{dT}{T} = 0 \qquad (5)$$

From the speed of sound relation, $$c^2 = kRT = (k-1)c_p T = kP/\rho \qquad (6)$$

Combining Eqs. (3) and (5), $$\frac{dP}{P} - \frac{dT}{T} + \frac{dA}{A} + \frac{dV}{V} = 0 \qquad (7)$$

Combining Eqs. (4) and (6), $$\frac{dP}{\rho} = \frac{dP}{kP/c^2} = -VdV$$

or, $$\frac{dP}{P} = -\frac{k}{C^2}V\,dV = -k\frac{V^2}{C^2}\frac{dV}{V} = -k\text{Ma}^2\frac{dV}{V} \qquad (8)$$

Combining Eqs. (2) and (6), $$dT = dT_0 - V\frac{dV}{c_p}$$

or, $$\frac{dT}{T} = \frac{dT_0}{T} - \frac{V^2}{C_pT}\frac{dV}{V} = \frac{dT}{T} = \frac{dT_0}{T} - \frac{V^2}{C^2/(k-1)}\frac{dV}{V} = \frac{dT_0}{T} - (k-1)\text{Ma}^2\frac{dV}{V} \qquad (9)$$

Combining Eqs. (7), (8), and (9), $$-(k-1)\text{Ma}^2\frac{dV}{V} - \frac{dT_0}{T} + (k-1)\text{Ma}^2\frac{dV}{V} + \frac{dA}{A} + \frac{dV}{V} = 0$$

or, $$\frac{dT_0}{T} = \frac{dA}{A} + \left[-k\text{Ma}^2 + (k-1)\text{Ma}^2 + 1\right]\frac{dV}{V}$$

Thus, $$\boxed{\frac{dT_0}{T} = \frac{dA}{A} + (1-\text{Ma}^2)\frac{dV}{V}} \qquad (10)$$

Differentiating the steady-flow energy equation $q = h_{02} - h_{01} = c_p(T_{02} - T_{01})$

$$\delta q = c_p dT_0 \qquad (11)$$

Eq. (11) relates the stagnation temperature change $dT_0$ to the net heat transferred to the fluid. Eq. (10) relates the velocity changes to area changes $dA$, and the stagnation temperature change $dT_0$ or the heat transferred.

(*a*) When Ma < 1 (subsonic flow), **the fluid accelerates if the duct converges** ($dA < 0$) **or the fluid is heated** ($dT_0 > 0$ or $\delta q > 0$). **The fluid decelerates if the duct converges** ($dA < 0$) **or the fluid is cooled** ($dT_0 < 0$ or $\delta q < 0$).

(*b*) When Ma > 1 (supersonic flow), **the fluid accelerates if the duct diverges** ($dA > 0$) **or the fluid is cooled** ($dT_0 < 0$ or $\delta q < 0$). **The fluid decelerates if the duct converges** ($dA < 0$) **or the fluid is heated** ($dT_0 > 0$ or $\delta q > 0$).

***Discussion*** Some of these results are not intuitively obvious, but come about by satisfying the conservation equations.

**12-139**

**Solution** A Pitot-static probe measures the difference between the static and stagnation pressures for a subsonic airplane. The speed of the airplane and the flight Mach number are to be determined.

***Assumptions*** **1** Air is an ideal gas with a constant specific heat ratio. **2** The stagnation process is isentropic.

***Properties*** The properties of air are $R$ = 0.287 kJ/kg·K and $k$ = 1.4.

***Analysis*** The stagnation pressure of air at the specified conditions is

$$P_0 = P + \Delta P = 70.109 + 22 = 92.109\text{ kPa}$$

Then,

$$\frac{P_0}{P} = \left(1 + \frac{(k-1)\text{Ma}^2}{2}\right)^{k/k-1} \longrightarrow \frac{92.109}{70.109} = \left(1 + \frac{(1.4-1)\text{Ma}^2}{2}\right)^{1.4/0.4}$$

It yields **Ma = 0.637**

The speed of sound in air at the specified conditions is

$$c = \sqrt{kRT} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(268.65\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 328.5\text{ m/s}$$

Thus,

$$V = \text{Ma} \times c = (0.637)(328.5\text{ m/s}) = \mathbf{209\ m/s}$$

***Discussion*** Note that the flow velocity can be measured in a simple and accurate way by simply measuring pressure.

---

**12-140**

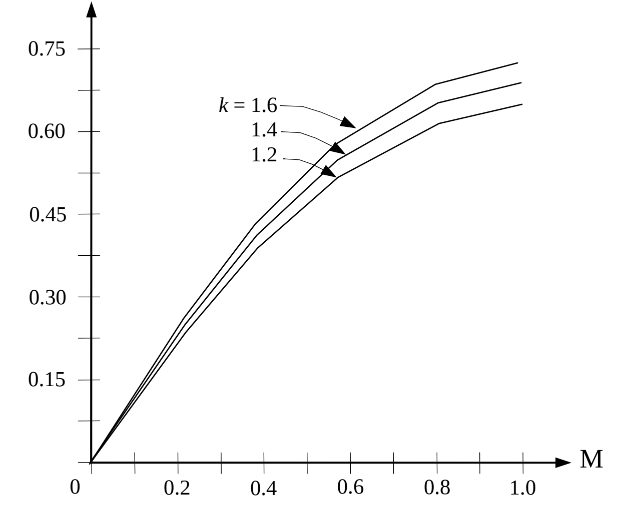
**Solution** The mass flow parameter $\dot{m}\sqrt{RT_0}/(AP_0)$ versus the Mach number for $k$ = 1.2, 1.4, and 1.6 in the range of $0 \le \text{Ma} \le 1$ is to be plotted.

***Analysis*** The mass flow rate parameter $(\dot{m}\sqrt{RT_0})/P_0 A$ can be expressed as

$$\frac{\dot{m}\sqrt{RT_0}}{P_0 A} = \text{Ma}\sqrt{k}\left(\frac{2}{2+(k-1)M^2}\right)^{(k+1)/2(k-1)}$$

Thus,

| Ma | **k = 1.2** | **k = 1.4** | **k = 1.6** |
|---|---|---|---|
| 0.0 | 0 | 0 | 0 |
| 0.1 | 0.1089 | 0.1176 | 0.1257 |
| 0.2 | 0.2143 | 0.2311 | 0.2465 |
| 0.3 | 0.3128 | 0.3365 | 0.3582 |
| 0.4 | 0.4015 | 0.4306 | 0.4571 |
| 0.5 | 0.4782 | 0.5111 | 0.5407 |
| 0.6 | 0.5411 | 0.5763 | 0.6077 |
| 0.7 | 0.5894 | 0.6257 | 0.6578 |
| 0.8 | 0.6230 | 0.6595 | 0.6916 |
| 0.9 | 0.6424 | 0.6787 | 0.7106 |
| 1.0 | 0.6485 | 0.6847 | 0.7164 |




***Discussion*** Note that the mass flow rate increases with increasing Mach number and specific heat ratio. It levels off at Ma = 1, and remains constant (choked flow).

---

**12-141**

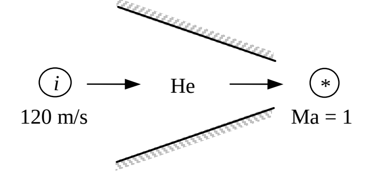
**Solution** Helium gas is accelerated in a nozzle. The pressure and temperature of helium at the location where Ma = 1 and the ratio of the flow area at this location to the inlet flow area are to be determined.

***Assumptions*** **1** Helium is an ideal gas with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic.

***Properties*** The properties of helium are $R$ = 2.0769 kJ/kg·K, $c_p$ = 5.1926 kJ/kg·K, and $k$ = 1.667.

***Analysis*** The properties of the fluid at the location where Ma = 1 are the critical properties, denoted by superscript *. We first determine the stagnation temperature and pressure, which remain constant throughout the nozzle since the flow is isentropic.

$$T_0 = T_i + \frac{V_i^2}{2c_p} = 500\text{ K} + \frac{(120\text{ m/s})^2}{2\times 5.1926\text{ kJ/kg}\cdot\text{K}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 501.4\text{ K}$$




and

$$P_0 = P_i\left(\frac{T_0}{T_i}\right)^{k/(k-1)} = (0.8\text{ MPa})\left(\frac{501.4\text{ K}}{500\text{ K}}\right)^{1.667/(1.667-1)} = 0.806\text{ MPa}$$

The Mach number at the nozzle exit is given to be Ma = 1. Therefore, the properties at the nozzle exit are the *critical properties* determined from

$$T^* = T_0\left(\frac{2}{k+1}\right) = (501.4\text{ K})\left(\frac{2}{1.667+1}\right) = \mathbf{376\text{ K}}$$

$$P^* = P_0\left(\frac{2}{k+1}\right)^{k/(k-1)} = (0.806\text{ MPa})\left(\frac{2}{1.667+1}\right)^{1.667/(1.667-1)} = \mathbf{0.393\text{ MPa}}$$

The speed of sound and the Mach number at the nozzle inlet are

$$c_i = \sqrt{kRT}_i = \sqrt{(1.667)(2.0769\text{ kJ/kg}\cdot\text{K})(500\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 1316\text{ m/s}$$

$$\text{Ma}_i = \frac{V_i}{c_i} = \frac{120\text{ m/s}}{1316\text{ m/s}} = 0.0912$$

The ratio of the entrance-to-throat area is

$$\frac{A_i}{A^*} = \frac{1}{\text{Ma}_i}\left[\left(\frac{2}{k+1}\right)\left(1+\frac{k-1}{2}\text{Ma}_i^2\right)\right]^{(k+1)/[2(k-1)]}$$

$$= \frac{1}{0.0912}\left[\left(\frac{2}{1.667+1}\right)\left(1+\frac{1.667-1}{2}(0.0912)^2\right)\right]^{2.667/(2\times 0.667)}$$

$$= 6.20$$

Then the ratio of the throat area to the entrance area becomes

$$\frac{A^*}{A_i} = \frac{1}{6.20} = \mathbf{0.161}$$

***Discussion*** The compressible flow functions are essential tools when determining the proper shape of the compressible flow duct.

**12-142**

**Solution** Helium gas enters a nozzle with negligible velocity, and is accelerated in a nozzle. The pressure and temperature of helium at the location where Ma = 1 and the ratio of the flow area at this location to the inlet flow area are to be determined.

***Assumptions*** **1** Helium is an ideal gas with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic. **3** The entrance velocity is negligible.

***Properties*** The properties of helium are $R$ = 2.0769 kJ/kg·K, $c_p$ = 5.1926 kJ/kg·K, and $k$ = 1.667.

***Analysis*** We treat helium as an ideal gas with $k$ = 1.667. The properties of the fluid at the location where Ma = 1 are the critical properties, denoted by superscript *.

The stagnation temperature and pressure in this case are identical to the inlet temperature and pressure since the inlet velocity is negligible. They remain constant throughout the nozzle since the flow is isentropic.

$$T_0 = T_i = 500 \text{ K}$$

$$P_0 = P_i = 0.8 \text{ MPa}$$

The Mach number at the nozzle exit is given to be Ma = 1. Therefore, the properties at the nozzle exit are the *critical properties* determined from

$$T^* = T_0\left(\frac{2}{k+1}\right) = (500\text{ K})\left(\frac{2}{1.667+1}\right) = \mathbf{375\ K}$$

$$P^* = P_0\left(\frac{2}{k+1}\right)^{k/(k-1)} = (0.8\text{ MPa})\left(\frac{2}{1.667+1}\right)^{1.667/(1.667-1)} = \mathbf{0.390\ MPa}$$



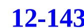
The ratio of the nozzle inlet area to the throat area is determined from

$$\frac{A_i}{A^*} = \frac{1}{\text{Ma}_i}\left[\left(\frac{2}{k+1}\right)\left(1+\frac{k-1}{2}\text{Ma}_i^2\right)\right]^{(k+1)/[2(k-1)]}$$

But the Mach number at the nozzle inlet is Ma = 0 since $V_i \cong 0$. Thus the ratio of the throat area to the nozzle inlet area is

$$\frac{A^*}{A_i} = \frac{1}{\infty} = \mathbf{0}$$

***Discussion*** The compressible flow functions are essential tools when determining the proper shape of the compressible flow duct.



## 12-143



**Solution** Air enters a converging nozzle. The mass flow rate, the exit velocity, the exit Mach number, and the exit pressure-stagnation pressure ratio versus the back pressure-stagnation pressure ratio for a specified back pressure range are to be calculated and plotted.

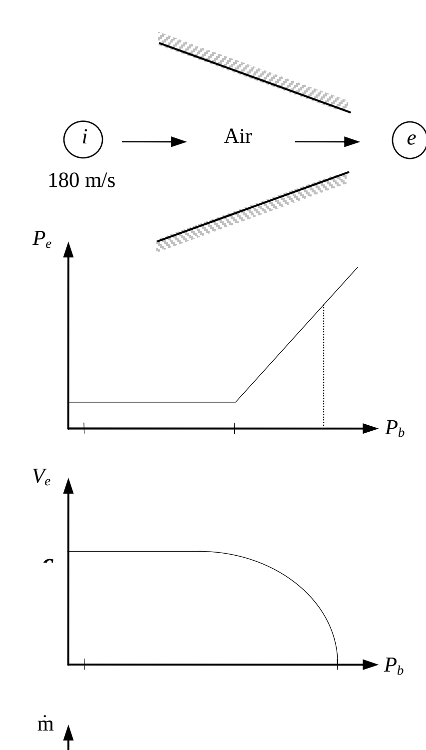
***Assumptions*** **1** Air is an ideal gas with constant specific heats at room temperature. **2** Flow through the nozzle is steady, one-dimensional, and isentropic.

***Properties*** The properties of air at room temperature are $R$ = 0.287 kJ/kg·K, $c_p$ = 1.005 kJ/kg·K, and $k$ = 1.4.

***Analysis*** We use EES to tabulate and plot the results. The stagnation properties remain constant throughout the nozzle since the flow is isentropic. They are determined from

$$T_0 = T_i + \frac{V_i^2}{2c_p} = 400\text{ K} + \frac{(180\text{ m/s})^2}{2\times 1.005\text{ kJ/kg}\cdot\text{K}}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 416.1\text{ K}$$

$$P_0 = P_i\left(\frac{T_0}{T_i}\right)^{k/(k-1)} = (900\text{ kPa})\left(\frac{416.1\text{ K}}{400\text{ K}}\right)^{1.4/(1.4-1)} = 1033.3\text{ kPa}$$




The critical pressure is determined to be

$$P^* = P_0\left(\frac{2}{k+1}\right)^{k/(k-1)} = (1033.3\text{ kPa})\left(\frac{2}{1.4+1}\right)^{1.4/0.4} = 545.9\text{ kPa}$$

Then the pressure at the exit plane (throat) is

$P_e = P_b$ for $P_b \geq 545.9$ kPa

$P_e = P^* = 545.9$ kPa for $P_b < 545.9$ kPa (choked flow)

Thus the back pressure does not affect the flow when $100 < P_b < 545.9$ kPa. For a specified exit pressure $P_e$, the temperature, velocity, and mass flow rate are

Temperature $$T_e = T_0\left(\frac{P_e}{P_0}\right)^{(k-1)/k} = (416.1\text{ K})\left(\frac{\text{P}_\text{e}}{1033.3}\right)^{0.4/1.4}$$

Velocity $$V = \sqrt{2c_p(T_0 - T_e)} = \sqrt{2(1.005\text{ kJ/kg}\cdot\text{K})(416.1 - T_e)\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)}$$

Speed of sound $$c_e = \sqrt{kRT}_e = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)}$$

Mach number $$\text{Ma}_e = V_e / c_e$$

Density $$\rho_e = \frac{P_e}{RT_e} = \frac{P_e}{(0.287\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})T_e}$$

Mass flow rate $$\dot{m} = \rho_e V_e A_e = \rho_e V_e (0.001\text{ m}^2)$$

| $P_b$, kPa | $P_b$, $P_0$ | $P_e$, kPa | $P_b$, $P_0$ | $T_e$, K | $V_e$, m/s | Ma | $\rho_e$, kg/m$^3$ | $\dot{m}$, kg / s |
|---|---|---|---|---|---|---|---|---|
| 900 | 0.871 | 900 | 0.871 | 400.0 | 180.0 | 0.45 | 7.840 | 0 |
| 800 | 0.774 | 800 | 0.774 | 386.8 | 162.9 | 0.41 | 7.206 | 1.174 |
| 700 | 0.677 | 700 | 0.677 | 372.3 | 236.0 | 0.61 | 6.551 | 1.546 |
| 600 | 0.581 | 600 | 0.581 | 356.2 | 296.7 | 0.78 | 5.869 | 1.741 |
| 545.9 | 0.528 | 545.9 | 0.528 | 333.3 | 366.2 | 1.00 | 4.971 | 1.820 |
| 500 | 0.484 | 545.9 | 0.528 | 333.2 | 366.2 | 1.00 | 4.971 | 1.820 |
| 400 | 0.387 | 545.9 | 0.528 | 333.3 | 366.2 | 1.00 | 4.971 | 1.820 |
| 300 | 0.290 | 545.9 | 0.528 | 333.3 | 366.2 | 1.00 | 4.971 | 1.820 |
| 200 | 0.194 | 545.9 | 0.528 | 333.3 | 366.2 | 1.00 | 4.971 | 1.820 |
| 100 | 0.097 | 545.9 | 0.528 | 333.3 | 366.2 | 1.00 | 4.971 | 1.820 |

***Discussion*** Once the back pressure drops below 545.0 kPa, the flow is choked, and $\dot{m}$ remains constant from then on.

**12-144** 

**Solution** Steam enters a converging nozzle. The exit pressure, the exit velocity, and the mass flow rate versus the back pressure for a specified back pressure range are to be plotted.

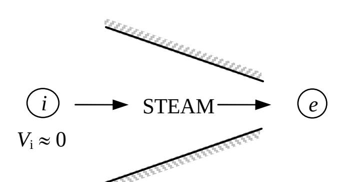
***Assumptions*** **1** Steam is to be treated as an ideal gas with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic. **3** The nozzle is adiabatic.

***Properties*** The ideal gas properties of steam are $R$ = 0.462 kJ/kg.K, $c_p$ = 1.872 kJ/kg.K, and $k$ = 1.3.

***Analysis*** We use EES to solve the problem. The stagnation properties in this case are identical to the inlet properties since the inlet velocity is negligible. Since the flow is isentropic, they remain constant throughout the nozzle,

$P_0 = P_i = 6$ MPa and $T_0 = T_i = 700$ K

The critical pressure is determined to be



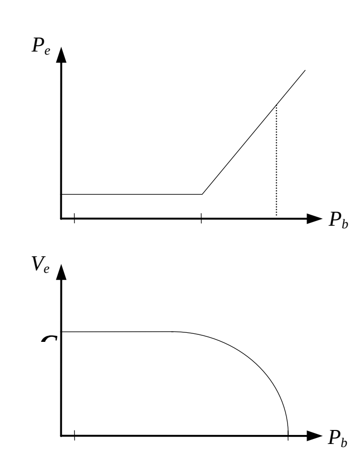

$$P^* = P_0\left(\frac{2}{k+1}\right)^{k/(k-1)} = (6\text{ MPa})\left(\frac{2}{1.3+1}\right)^{1.3/0.3} = 3.274\text{ MPa}$$

Then the pressure at the exit plane (throat) is

$P_e = P_b$ for $P_b \geq 3.274$ MPa

$P_e = P^* = 3.274$ MPa for $P_b < 3.274$ MPa (choked flow)

Thus the back pressure does not affect the flow when $3 < P_b < 3.274$ MPa. For a specified exit pressure $P_e$, the temperature, velocity, and mass flow rate are



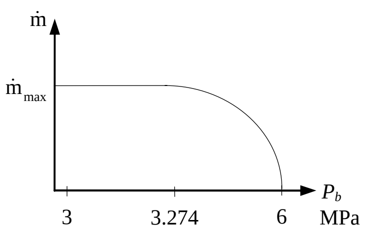

Temperature $$T_e = T_0\left(\frac{P_e}{P_0}\right)^{(k-1)/k} = (700\text{ K})\left(\frac{P_e}{6}\right)^{0.3/1.3}$$

Velocity $$V = \sqrt{2c_p(T_0 - T_e)} = \sqrt{2(1.872\text{ kJ/kg}\cdot\text{K})(700 - T_e)\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)}$$

Density $$\rho_e = \frac{P_e}{RT_e} = \frac{P_e}{(0.462\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})T_e}$$

Mass flow rate $$\dot{m} = \rho_e V_e A_e = \rho_e V_e (0.0008\text{ m}^2)$$

The results of the calculations are tabulated as follows:




| $P_b$, MPa | $P_e$, MPa | $T_e$, K | $V_e$, m/s | $\rho_e$, kg/m³ | $\dot{m}$, kg/s |
|---|---|---|---|---|---|
| 6.0 | 6.0 | 700 | 0 | 18.55 | 0 |
| 5.5 | 5.5 | 686.1 | 228.1 | 17.35 | 3.166 |
| 5.0 | 5.0 | 671.2 | 328.4 | 16.12 | 4.235 |
| 4.5 | 4.5 | 655.0 | 410.5 | 14.87 | 4.883 |
| 4.0 | 4.0 | 637.5 | 483.7 | 13.58 | 5.255 |
| 3.5 | 3.5 | 618.1 | 553.7 | 12.26 | 5.431 |
| 3.274 | 3.274 | 608.7 | 584.7 | 11.64 | 5.445 |
| 3.0 | 3.274 | 608.7 | 584.7 | 11.64 | 5.445 |

***Discussion*** Once the back pressure drops below 3.274 MPa, the flow is choked, and $\dot{m}$ remains constant from then on.

**12-145**

**Solution** An expression for the ratio of the stagnation pressure after a shock wave to the static pressure before the shock wave as a function of $k$ and the Mach number upstream of the shock wave is to be found.

***Analysis*** The relation between $P_1$ and $P_2$ is

$$\frac{P_2}{P_1} = \frac{1+k\text{Ma}_2^2}{1+k\text{Ma}_1^2} \longrightarrow P_2 = P_1\left(\frac{1+k\text{Ma}_1^2}{1+k\text{Ma}_2^2}\right)$$

We substitute this into the isentropic relation

$$\frac{P_{02}}{P_2} = \left(1+\left(k-1\right)\text{Ma}_2^2/2\right)^{k/(k-1)}$$

which yields

$$\frac{P_{02}}{P_1} = \left(\frac{1+k\text{Ma}_1^2}{1+k\text{Ma}_2^2}\right)\left(1+(k-1)\text{Ma}_2^2/2\right)^{k/(k-1)}$$

where

$$\text{Ma}_2^2 = \frac{\text{Ma}_1^2 + 2/(k-1)}{2k\text{Ma}_2^2/(k-1)-1}$$

Substituting,

$$\boxed{\frac{P_{02}}{P_1} = \left(\frac{(1+k\text{Ma}_1^2)(2k\text{Ma}_1^2-k+1)}{k\text{Ma}_1^2(k+1)-k+3}\right)\left(1+\frac{(k-1)\text{Ma}_1^2/2+1}{2k\text{Ma}_1^2/(k-1)-1}\right)^{k/(k-1)}}$$

***Discussion*** Similar manipulations of the equations can be performed to get the ratio of other parameters across a shock.

**12-146**

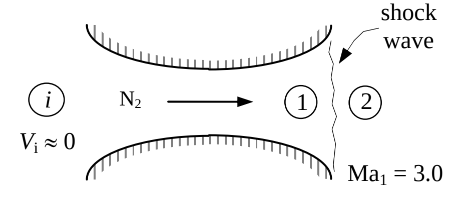
**Solution** Nitrogen entering a converging-diverging nozzle experiences a normal shock. The pressure, temperature, velocity, Mach number, and stagnation pressure downstream of the shock are to be determined. The results are to be compared to those of air under the same conditions.

***Assumptions*** **1** Nitrogen is an ideal gas with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic. **3** The nozzle is adiabatic.

***Properties*** The properties of nitrogen are $R$ = 0.297 kJ/kg·K and $k$ = 1.4.

***Analysis*** The inlet stagnation properties in this case are identical to the inlet properties since the inlet velocity is negligible. Assuming the flow before the shock to be isentropic,

$$P_{01} = P_i = 700 \text{ kPa}$$
$$T_{01} = T_i = 300 \text{ K}$$




Then,

$$T_1 = T_{01}\left(\frac{2}{2+(k-1)\text{Ma}_1^2}\right) = (300\text{ K})\left(\frac{2}{2+(1.4-1)3^2}\right) = 107.1\text{ K}$$

and

$$P_1 = P_{01}\left(\frac{T_1}{T_{01}}\right)^{k/(k-1)} = (700\text{ kPa})\left(\frac{107.1}{300}\right)^{1.4/0.4} = 19.06\text{ kPa}$$

The fluid properties after the shock (denoted by subscript 2) are related to those before the shock through the functions listed in Table A-14. For $\text{Ma}_1$ = 3.0 we read

$$\text{Ma}_2 = 0.4752 \cong \mathbf{0.475}, \quad \frac{P_{02}}{P_{01}} = 0.32834, \quad \frac{P_2}{P_1} = 10.333, \quad \text{and} \quad \frac{T_2}{T_1} = 2.679$$

Then the stagnation pressure $P_{02}$, static pressure $P_2$, and static temperature $T_2$, are determined to be

$$P_{02} = 0.32834P_{01} = (0.32834)(700\text{ kPa}) = \mathbf{230\ kPa}$$
$$P_2 = 10.333P_1 = (10.333)(19.06\text{ kPa}) = \mathbf{197\ kPa}$$
$$T_2 = 2.679T_1 = (2.679)(107.1\text{ K}) = \mathbf{287\ K}$$

The velocity after the shock can be determined from $V_2 = \text{Ma}_2 c_2$, where $c_2$ is the speed of sound at the exit conditions after the shock,

$$V_2 = \text{Ma}_2 c_2 = \text{Ma}_2\sqrt{kRT_2} = (0.4752)\sqrt{(1.4)(0.297\text{ kJ/kg}\cdot\text{K})(287\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = \mathbf{164\ m/s}$$

***Discussion*** For *air* at specified conditions $k$ = 1.4 (same as nitrogen) and $R$ = 0.287 kJ/kg·K. Thus the only quantity which will be different in the case of air is the velocity after the normal shock, which happens to be 161.3 m/s.

**12-147**

**Solution** The diffuser of an aircraft is considered. The static pressure rise across the diffuser and the exit area are to be determined.

***Assumptions*** **1** Air is an ideal gas with constant specific heats at room temperature. **2** Flow through the diffuser is steady, one-dimensional, and isentropic. **3** The diffuser is adiabatic.

***Properties*** Air properties at room temperature are $R$ = 0.287 kJ/kg·K, $c_p$ = 1.005 kJ/kg·K, and $k$ = 1.4.

***Analysis*** The inlet velocity is

$$V_1 = \text{Ma}_1 c_1 = M_1\sqrt{kRT_1} = (0.8)\sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(242.7\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 249.8\text{ m/s}$$

Then the stagnation temperature and pressure at the diffuser inlet become




$$T_{01} = T_1 + \frac{V_1^2}{2c_p} = 242.7 + \frac{(249.8\text{ m/s})^2}{2(1.005\text{ kJ/kg}\cdot\text{K})}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 273.7\text{ K}$$

$$P_{01} = P_1\left(\frac{T_{01}}{T_1}\right)^{k/(k-1)} = (41.1\text{ kPa})\left(\frac{273.7\text{ K}}{242.7\text{ K}}\right)^{1.4/(1.4-1)} = 62.6\text{ kPa}$$

For an adiabatic diffuser, the energy equation reduces to $h_{01} = h_{02}$. Noting that $h = c_pT$ and the specific heats are assumed to be constant, we have

$$T_{01} = T_{02} = T_0 = 273.7\text{ K}$$

The isentropic relation between states 1 and 02 gives

$$P_{02} = P_{02} = P_1\left(\frac{T_{02}}{T_1}\right)^{k/(k-1)} = (41.1\text{ kPa})\left(\frac{273.72\text{ K}}{242.7\text{ K}}\right)^{1.4/(1.4-1)} = 62.61\text{ kPa}$$

The exit velocity can be expressed as

$$V_2 = \text{Ma}_2 c_2 = \text{Ma}_2\sqrt{kRT_2} = (0.3)\sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})\,T_2\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 6.01\sqrt{T_2}$$

Thus $$T_2 = T_{02} - \frac{V_2^2}{2c_p} = (273.7) - \frac{6.01^2\,\text{T}_2\text{ m}^2/\text{s}^2}{2(1.005\text{ kJ/kg}\cdot\text{K})}\left(\frac{1\text{ kJ/kg}}{1000\text{ m}^2/\text{s}^2}\right) = 268.9\text{ K}$$

Then the static exit pressure becomes

$$P_2 = P_{02}\left(\frac{T_2}{T_{02}}\right)^{k/(k-1)} = (62.61\text{ kPa})\left(\frac{268.9\text{ K}}{273.7\text{ K}}\right)^{1.4/(1.4-1)} = 58.85\text{ kPa}$$

Thus the static pressure rise across the diffuser is

$$\Delta P = P_2 - P_1 = 58.85 - 41.1 = \mathbf{17.8\ kPa}$$

Also, $$\rho_2 = \frac{P_2}{RT_2} = \frac{58.85\text{ kPa}}{(0.287\text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(268.9\text{ K})} = 0.7626\text{ kg/m}^3$$

$$V_2 = 6.01\sqrt{T_2} = 6.01\sqrt{268.9} = 98.6\text{ m/s}$$

Thus $$A_2 = \frac{\dot m}{\rho_2 V_2} = \frac{65\text{ kg/s}}{(0.7626\text{ kg/m}^3)(98.6\text{ m/s})} = \mathbf{0.864\ m^2}$$

***Discussion*** The pressure rise in actual diffusers will be lower because of the irreversibilities. However, flow through well-designed diffusers is very nearly isentropic.

**12-148**

**Solution** Helium gas is accelerated in a nozzle isentropically. For a specified mass flow rate, the throat and exit areas of the nozzle are to be determined.

***Assumptions*** **1** Helium is an ideal gas with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic. **3** The nozzle is adiabatic.

***Properties*** The properties of helium are $R$ = 2.0769 kJ/kg.K, $c_p$ = 5.1926 kJ/kg.K, and $k$ = 1.667.

***Analysis*** The inlet stagnation properties in this case are identical to the inlet properties since the inlet velocity is negligible,

$$T_{01} = T_1 = 500 \text{ K}$$

$$P_{01} = P_1 = 1.0 \text{ MPa}$$

The flow is assumed to be isentropic, thus the stagnation temperature and pressure remain constant throughout the nozzle,



$$T_{02} = T_{01} = 500 \text{ K}$$

$$P_{02} = P_{01} = 1.0 \text{ MPa}$$

The critical pressure and temperature are determined from

$$T^* = T_0\left(\frac{2}{k+1}\right) = (500 \text{ K})\left(\frac{2}{1.667+1}\right) = 375.0 \text{ K}$$

$$P^* = P_0\left(\frac{2}{k+1}\right)^{k/(k-1)} = (1.0 \text{ MPa})\left(\frac{2}{1.667+1}\right)^{1.667/(1.667-1)} = 0.487 \text{ MPa}$$

$$\rho^* = \frac{P^*}{RT^*} = \frac{487 \text{ kPa}}{(2.0769 \text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(375 \text{ K})} = 0.625 \text{ kg/m}^3$$

$$V^* = c^* = \sqrt{kRT^*} = \sqrt{(1.667)(2.0769 \text{ kJ/kg}\cdot\text{K})(375 \text{ K})\left(\frac{1000 \text{ m}^2/\text{s}^2}{1 \text{ kJ/kg}}\right)} = 1139.4 \text{ m/s}$$

Thus the throat area is

$$A^* = \frac{\dot{m}}{\rho^* V^*} = \frac{0.25 \text{ kg/s}}{(0.625 \text{ kg/m}^3)(1139.4 \text{ m/s})} = 3.51\times10^{-4} \text{ m}^2 = \mathbf{3.51\ cm^2}$$

At the nozzle exit the pressure is $P_2$ = 0.1 MPa. Then the other properties at the nozzle exit are determined to be

$$\frac{P_0}{P_2} = \left(1+\frac{k-1}{2}\text{Ma}_2^2\right)^{k/(k-1)} \longrightarrow \frac{1.0 \text{ MPa}}{0.1 \text{ MPa}} = \left(1+\frac{1.667-1}{2}\text{Ma}_2^2\right)^{1.667/0.667}$$

It yields $\text{Ma}_2$ = 2.130, which is greater than 1. Therefore, the nozzle must be converging-diverging.

$$T_2 = T_0\left(\frac{2}{2+(k-1)\text{Ma}_2^2}\right) = (500 \text{ K})\left(\frac{2}{2+(1.667-1)\times 2.13^2}\right) = 199.0 \text{ K}$$

$$\rho_2 = \frac{P_2}{RT_2} = \frac{100 \text{ kPa}}{(2.0769 \text{ kPa}\cdot\text{m}^3/\text{kg}\cdot\text{K})(199 \text{ K})} = 0.242 \text{ kg/m}^3$$

$$V_2 = \text{Ma}_2 c_2 = \text{Ma}_2\sqrt{kRT_2} = (2.13)\sqrt{(1.667)(2.0769 \text{ kJ/kg}\cdot\text{K})(199 \text{ K})\left(\frac{1000 \text{ m}^2/\text{s}^2}{1 \text{ kJ/kg}}\right)} = 1768.0 \text{ m/s}$$

Thus the exit area is

$$A_2 = \frac{\dot{m}}{\rho_2 V_2} = \frac{0.25 \text{ kg/s}}{(0.242 \text{ kg/m}^3)(1768 \text{ m/s})} = 5.84\times10^{-4} \text{ m}^2 = \mathbf{5.84\ cm^2}$$

***Discussion*** Flow areas in actual nozzles would be somewhat larger to accommodate the irreversibilities.

**12-149E**

**Solution** Helium gas is accelerated in a nozzle. For a specified mass flow rate, the throat and exit areas of the nozzle are to be determined for the cases of isentropic and 97% efficient nozzles.

***Assumptions*** **1** Helium is an ideal gas with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic. **3** The nozzle is adiabatic.

***Properties*** The properties of helium are $R$ = 0.4961 Btu/lbm·R = 2.6809 psia·ft$^3$/lbm·R, $c_p$ = 1.25 Btu/lbm·R, and $k$ = 1.667.

***Analysis*** The inlet stagnation properties in this case are identical to the inlet properties since the inlet velocity is negligible,

$$T_{01} = T_1 = 900 \text{ R}$$

$$P_{01} = P_1 = 150 \text{ psia}$$

The flow is assumed to be isentropic, thus the stagnation temperature and pressure remain constant throughout the nozzle,

$$T_{02} = T_{01} = 900 \text{ R}$$

$$P_{02} = P_{01} = 150 \text{ psia}$$

1 He * 2
$V_i \approx 0$

The critical pressure and temperature are determined from

$$T^* = T_0\left(\frac{2}{k+1}\right) = (900\text{ R})\left(\frac{2}{1.667+1}\right) = 674.9\text{ R}$$

$$P^* = P_0\left(\frac{2}{k+1}\right)^{k/(k-1)} = (150\text{ psia})\left(\frac{2}{1.667+1}\right)^{1.667/(1.667-1)} = 73.1\text{ psia}$$

$$\rho^* = \frac{P^*}{RT^*} = \frac{73.1\text{ psia}}{(2.6809\text{ psia}\cdot\text{ft}^3/\text{lbm}\cdot\text{R})(674.9\text{ R})} = 0.0404\text{ 1bm/ft}^3$$

$$V^* = c^* = \sqrt{kRT^*} = \sqrt{(1.667)(0.4961\text{ Btu/lbm}\cdot\text{R})(674.9\text{ R})\left(\frac{25{,}037\text{ ft}^2/\text{s}^2}{1\text{ Btu/1bm}}\right)} = 3738\text{ ft/s}$$

and $$A^* = \frac{\dot m}{\rho^* V^*} = \frac{0.2\text{ 1bm/s}}{(0.0404\text{ 1bm/ft}^3)(3738\text{ ft/s})} = \mathbf{0.00132\ ft^2}$$

At the nozzle exit the pressure is $P_2$ = 15 psia. Then the other properties at the nozzle exit are determined to be

$$\frac{p_0}{p_2} = \left(1+\frac{k-1}{2}\text{Ma}_2^2\right)^{k/(k-1)} \longrightarrow \frac{150\text{ psia}}{15\text{ psia}} = \left(1+\frac{1.667-1}{2}\text{Ma}_2^2\right)^{1.667/0.667}$$

It yields $\text{Ma}_2$ = 2.130, which is greater than 1. Therefore, the nozzle must be converging-diverging.

$$T_2 = T_0\left(\frac{2}{2+(k-1)\text{Ma}_2^2}\right) = (900\text{ R})\left(\frac{2}{2+(1.667-1)\times 2.13^2}\right) = 358.1\text{ R}$$

$$\rho_2 = \frac{P_2}{RT_2} = \frac{15\text{ psia}}{(2.6809\text{ psia}\cdot\text{ft}^3/\text{lbm}\cdot\text{R})(358.1\text{ R})} = 0.0156\text{ 1bm/ft}^3$$

$$V_2 = \text{Ma}_2 c_2 = \text{Ma}_2\sqrt{kRT_2} = (2.13)\sqrt{(1.667)(0.4961\text{ Btu/lbm}\cdot\text{R})(358.1\text{ R})\left(\frac{25{,}037\text{ ft}^2/\text{s}^2}{1\text{ Btu/1bm}}\right)} = 5800\text{ ft/s}$$

Thus the exit area is

$$A_2 = \frac{\dot m}{\rho_2 V_2} = \frac{0.2\text{ lbm/s}}{(0.0156\text{ lbm/ft}^3)(5800\text{ ft/s})} = \mathbf{0.00221\ ft^2}$$

***Discussion*** Flow areas in actual nozzles would be somewhat larger to accommodate the irreversibilities.

**12-150** [*Also solved using EES on enclosed DVD*]

**Solution** Using the compressible flow relations, the one-dimensional compressible flow functions are to be evaluated and tabulated as in Table A-13 for an ideal gas with $k$ = 1.667.

***Properties*** The specific heat ratio of the ideal gas is given to be $k$ = 1.667.

***Analysis*** The compressible flow functions listed below are expressed in EES and the results are tabulated.

$$\mathrm{Ma}^* = \mathrm{Ma}\sqrt{\frac{k+1}{2+(k-1)\mathrm{Ma}^2}} \qquad \frac{A}{A^*} = \frac{1}{\mathrm{Ma}}\left[\left(\frac{2}{k+1}\right)\left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)\right]^{0.5(k+1)/(k-1)}$$

$$\frac{P}{P_0} = \left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)^{-k/(k-1)} \qquad \frac{\rho}{\rho_0} = \left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)^{-1/(k-1)}$$

$$\frac{T}{T_0} = \left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)^{-1}$$

```
k=1.667
PP0=(1+(k-1)*M^2/2)^(-k/(k-1))
TT0=1/(1+(k-1)*M^2/2)
DD0=(1+(k-1)*M^2/2)^(-1/(k-1))
Mcr=M*SQRT((k+1)/(2+(k-1)*M^2))
AAcr=((2/(k+1))*(1+0.5*(k-1)*M^2))^(0.5*(k+1)/(k-1))/M
```

| Ma | $\mathrm{Ma}^*$ | $A/A^*$ | $P/P_0$ | $\rho/\rho_0$ | $T/T_0$ |
|---|---|---|---|---|---|
| 0.0 | 0 | ∞ | 1.0000 | 1.0000 | 1.0000 |
| 0.1 | 0.1153 | 5.6624 | 0.9917 | 0.9950 | 0.9967 |
| 0.2 | 0.2294 | 2.8879 | 0.9674 | 0.9803 | 0.9868 |
| 0.3 | 0.3413 | 1.9891 | 0.9288 | 0.9566 | 0.9709 |
| 0.4 | 0.4501 | 1.5602 | 0.8782 | 0.9250 | 0.9493 |
| 0.5 | 0.5547 | 1.3203 | 0.8186 | 0.8869 | 0.9230 |
| 0.6 | 0.6547 | 1.1760 | 0.7532 | 0.8437 | 0.8928 |
| 0.7 | 0.7494 | 1.0875 | 0.6850 | 0.7970 | 0.8595 |
| 0.8 | 0.8386 | 1.0351 | 0.6166 | 0.7482 | 0.8241 |
| 0.9 | 0.9222 | 1.0081 | 0.5501 | 0.6987 | 0.7873 |
| 1.0 | 1.0000 | 1.0000 | 0.4871 | 0.6495 | 0.7499 |
| 1.2 | 1.1390 | 1.0267 | 0.3752 | 0.5554 | 0.6756 |
| 1.4 | 1.2572 | 1.0983 | 0.2845 | 0.4704 | 0.6047 |
| 1.6 | 1.3570 | 1.2075 | 0.2138 | 0.3964 | 0.5394 |
| 1.8 | 1.4411 | 1.3519 | 0.1603 | 0.3334 | 0.4806 |
| 2.0 | 1.5117 | 1.5311 | 0.1202 | 0.2806 | 0.4284 |
| 2.2 | 1.5713 | 1.7459 | 0.0906 | 0.2368 | 0.3825 |
| 2.4 | 1.6216 | 1.9980 | 0.0686 | 0.2005 | 0.3424 |
| 2.6 | 1.6643 | 2.2893 | 0.0524 | 0.1705 | 0.3073 |
| 2.8 | 1.7007 | 2.6222 | 0.0403 | 0.1457 | 0.2767 |
| 3.0 | 1.7318 | 2.9990 | 0.0313 | 0.1251 | 0.2499 |
| 5.0 | 1.8895 | 9.7920 | 0.0038 | 0.0351 | 0.1071 |
| ∝ | 1.9996 | ∞ | 0 | 0 | 0 |

***Discussion*** The tabulated values are useful for quick calculations, but be careful – they apply only to one specific value of $k$, in this case $k$ = 1.667.

**12-151** **[*Also solved using EES on enclosed DVD*]**

**Solution** Using the normal shock relations, the normal shock functions are to be evaluated and tabulated as in Table A-14 for an ideal gas with $k$ = 1.667.

***Properties*** The specific heat ratio of the ideal gas is given to be $k$ = 1.667.

***Analysis*** The normal shock relations listed below are expressed in EES and the results are tabulated.

$$\mathrm{Ma}_2 = \sqrt{\frac{(k-1)\mathrm{Ma}_1^2 + 2}{2k\mathrm{Ma}_1^2 - k + 1}} \qquad \frac{P_2}{P_1} = \frac{1+k\mathrm{Ma}_1^2}{1+k\mathrm{Ma}_2^2} = \frac{2k\mathrm{Ma}_1^2 - k + 1}{k+1}$$

$$\frac{T_2}{T_1} = \frac{2+\mathrm{Ma}_1^2(k-1)}{2+\mathrm{Ma}_2^2(k-1)} \qquad \frac{\rho_2}{\rho_1} = \frac{P_2/P_1}{T_2/T_1} = \frac{(k+1)\mathrm{Ma}_1^2}{2+(k-1)\mathrm{Ma}_1^2} = \frac{V_1}{V_2},$$

$$\frac{P_{02}}{P_{01}} = \frac{\mathrm{Ma}_1}{\mathrm{Ma}_2}\left[\frac{1+\mathrm{Ma}_2^2(k-1)/2}{1+\mathrm{Ma}_1^2(k-1)/2}\right]^{\frac{k+1}{2(k-1)}} \qquad \frac{P_{02}}{P_1} = \frac{(1+k\mathrm{Ma}_1^2)[1+\mathrm{Ma}_2^2(k-1)/2]^{k/(k-1)}}{1+k\mathrm{Ma}_2^2}$$

```
k=1.667
My=SQRT((Mx^2+2/(k-1))/(2*Mx^2*k/(k-1)-1))
PyPx=(1+k*Mx^2)/(1+k*My^2)
TyTx=(1+Mx^2*(k-1)/2)/(1+My^2*(k-1)/2)
RyRx=PyPx/TyTx
P0yP0x=(Mx/My)*((1+My^2*(k-1)/2)/(1+Mx^2*(k-1)/2))^(0.5*(k+1)/(k-1))
P0yPx=(1+k*Mx^2)*(1+My^2*(k-1)/2)^(k/(k-1))/(1+k*My^2)
```

| $\mathrm{Ma}_1$ | $\mathrm{Ma}_2$ | $P_2/P_1$ | $\rho_2/\rho_1$ | $T_2/T_1$ | $P_{02}/P_{01}$ | $P_{02}/P_1$ |
|---|---|---|---|---|---|---|
| 1.0 | 1.0000 | 1.0000 | 1.0000 | 1.0000 | 1 | 2.0530 |
| 1.1 | 0.9131 | 1.2625 | 1.1496 | 1.0982 | 0.999 | 2.3308 |
| 1.2 | 0.8462 | 1.5500 | 1.2972 | 1.1949 | 0.9933 | 2.6473 |
| 1.3 | 0.7934 | 1.8626 | 1.4413 | 1.2923 | 0.9813 | 2.9990 |
| 1.4 | 0.7508 | 2.2001 | 1.5805 | 1.3920 | 0.9626 | 3.3838 |
| 1.5 | 0.7157 | 2.5626 | 1.7141 | 1.4950 | 0.938 | 3.8007 |
| 1.6 | 0.6864 | 2.9501 | 1.8415 | 1.6020 | 0.9085 | 4.2488 |
| 1.7 | 0.6618 | 3.3627 | 1.9624 | 1.7135 | 0.8752 | 4.7278 |
| 1.8 | 0.6407 | 3.8002 | 2.0766 | 1.8300 | 0.8392 | 5.2371 |
| 1.9 | 0.6227 | 4.2627 | 2.1842 | 1.9516 | 0.8016 | 5.7767 |
| 2.0 | 0.6070 | 4.7503 | 2.2853 | 2.0786 | 0.763 | 6.3462 |
| 2.1 | 0.5933 | 5.2628 | 2.3802 | 2.2111 | 0.7243 | 6.9457 |
| 2.2 | 0.5814 | 5.8004 | 2.4689 | 2.3493 | 0.6861 | 7.5749 |
| 2.3 | 0.5708 | 6.3629 | 2.5520 | 2.4933 | 0.6486 | 8.2339 |
| 2.4 | 0.5614 | 6.9504 | 2.6296 | 2.6432 | 0.6124 | 8.9225 |
| 2.5 | 0.5530 | 7.5630 | 2.7021 | 2.7989 | 0.5775 | 9.6407 |
| 2.6 | 0.5455 | 8.2005 | 2.7699 | 2.9606 | 0.5442 | 10.3885 |
| 2.7 | 0.5388 | 8.8631 | 2.8332 | 3.1283 | 0.5125 | 11.1659 |
| 2.8 | 0.5327 | 9.5506 | 2.8923 | 3.3021 | 0.4824 | 11.9728 |
| 2.9 | 0.5273 | 10.2632 | 2.9476 | 3.4819 | 0.4541 | 12.8091 |
| 3.0 | 0.5223 | 11.0007 | 2.9993 | 3.6678 | 0.4274 | 13.6750 |
| 4.0 | 0.4905 | 19.7514 | 3.3674 | 5.8654 | 0.2374 | 23.9530 |
| 5.0 | 0.4753 | 31.0022 | 3.5703 | 8.6834 | 0.1398 | 37.1723 |
| ∞ | 0.4473 | ∞ | 3.9985 | ∞ | 0 | ∞ |

***Discussion*** The tabulated values are useful for quick calculations, but be careful – they apply only to one specific value of $k$, in this case $k$ = 1.667.

**12-152**

**Solution** The critical temperature, pressure, and density of an equimolar mixture of oxygen and nitrogen for specified stagnation properties are to be determined.

***Assumptions*** Both oxygen and nitrogen are ideal gases with constant specific heats at room temperature.

***Properties*** The specific heat ratio and molar mass are $k$ = 1.395 and $M$ = 32 kg/kmol for oxygen, and $k$ = 1.4 and $M$ = 28 kg/kmol for nitrogen.

***Analysis*** The gas constant of the mixture is

$$M_m = y_{O_2} M_{O_2} + y_{N_2} M_{N_2} = 0.5 \times 32 + 0.5 \times 28 = 30 \text{ kg/kmol}$$

$$R_m = \frac{R_u}{M_m} = \frac{8.314 \text{ kJ/kmol} \cdot \text{K}}{30 \text{ kg/kmol}} = 0.2771 \text{ kJ/kg} \cdot \text{K}$$

The specific heat ratio is 1.4 for nitrogen, and nearly 1.4 for oxygen. Therefore, the specific heat ratio of the mixture is also 1.4. Then the critical temperature, pressure, and density of the mixture become

$$T^* = T_0 \left( \frac{2}{k+1} \right) = (800 \text{ K}) \left( \frac{2}{1.4+1} \right) = \mathbf{667 \text{ K}}$$

$$P^* = P_0 \left( \frac{2}{k+1} \right)^{k/(k-1)} = (500 \text{ kPa}) \left( \frac{2}{1.4+1} \right)^{1.4/(1.4-1)} = \mathbf{264 \text{ kPa}}$$

$$\rho^* = \frac{P^*}{RT^*} = \frac{264 \text{ kPa}}{(0.2771 \text{ kPa} \cdot \text{m}^3/\text{kg} \cdot \text{K})(667 \text{ K})} = \mathbf{1.43 \text{ kg/m}^3}$$

***Discussion*** If the specific heat ratios $k$ of the two gases were different, then we would need to determine the $k$ of the mixture from $k = C_{p,m}/C_{v,m}$ where the specific heats of the mixture are determined from

$$C_{p,m} = \text{mf}_{O_2} C_{p,O_2} + \text{mf}_{N_2} C_{p,N_2} = (y_{O_2} M_{O_2} / M_m) C_{p,O_2} + (y_{N_2} M_{N_2} / M_m) C_{p,N_2}$$
$$C_{v,m} = \text{mf}_{O_2} C_{v,O_2} + \text{mf}_{N_2} C_{v,N_2} = (y_{O_2} M_{O_2} / M_m) C_{v,O_2} + (y_{N_2} M_{N_2} / M_m) C_{v,N_2}$$

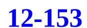
where mf is the mass fraction and $y$ is the mole fraction. In this case it would give

$$C_{p,m} = (0.5 \times 32/30) \times 0.918 + (0.5 \times 28/30) \times 1.039 = 0.974 \text{ kJ/kg.K}$$

$$C_{p,m} = (0.5 \times 32/30) \times 0.658 + (0.5 \times 28/30) \times 0.743 = 0.698 \text{ kJ/kg.K}$$

and

$k$ = 0.974/0.698 = 1.40



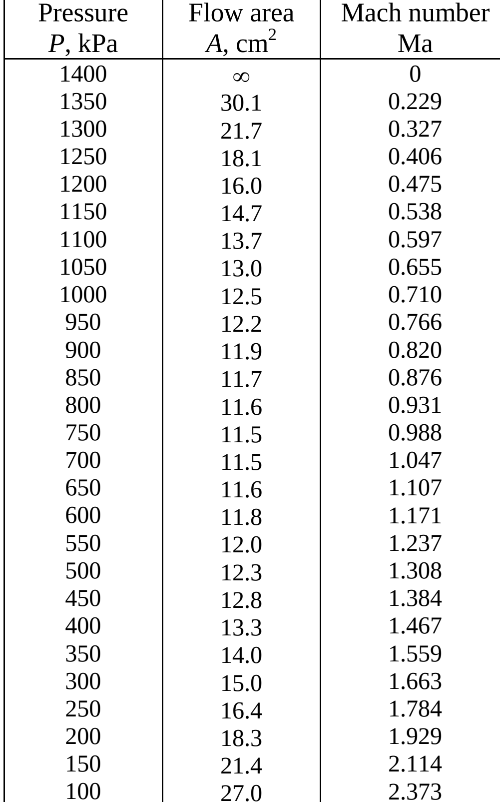
**12-153** 

**Solution** Using EES (or other) software, the shape of a converging-diverging nozzle is to be determined for specified flow rate and stagnation conditions. The nozzle and the Mach number are to be plotted.

***Assumptions*** **1** Air is an ideal gas with constant specific heats. **2** Flow through the nozzle is steady, one-dimensional, and isentropic. **3** The nozzle is adiabatic.

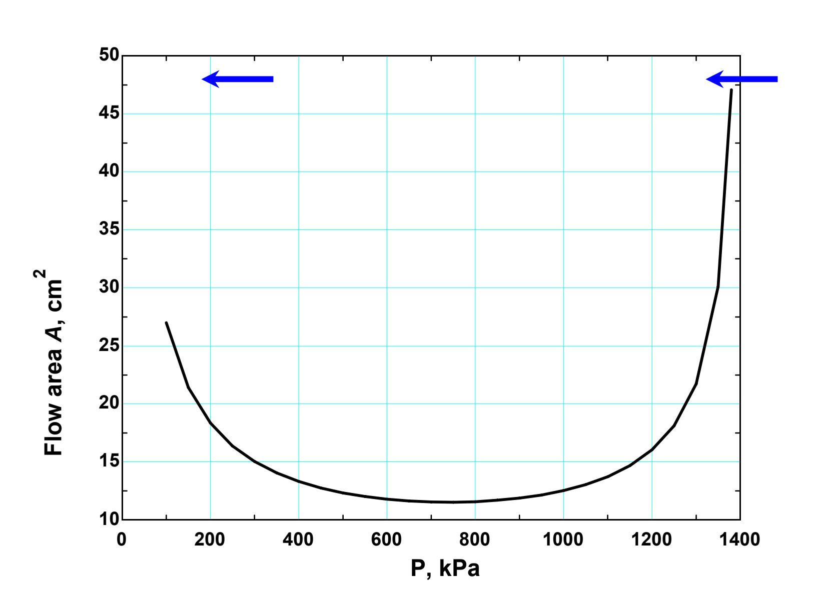
***Properties*** The specific heat ratio of air at room temperature is 1.4.

***Analysis*** The problem is solved using EES, and the results are tabulated and plotted below.

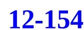


```
k=1.4
Cp=1.005 "kJ/kg.K"
R=0.287 "kJ/kg.K"
P0=1400 "kPa"
T0=200+273 "K"
m=3 "kg/s"
rho_0=P0/(R*T0)
rho=P/(R*T)
T=T0*(P/P0)^((k-1)/k)
V=SQRT(2*Cp*(T0-T)*1000)
A=m/(rho*V)*10000 "cm2"
C=SQRT(k*R*T*1000)
Ma=V/C
```

| Pressure *P*, kPa | Flow area *A*, cm$^2$ | Mach number Ma |
|---|---|---|
| 1400 | ∞ | 0 |
| 1350 | 30.1 | 0.229 |
| 1300 | 21.7 | 0.327 |
| 1250 | 18.1 | 0.406 |
| 1200 | 16.0 | 0.475 |
| 1150 | 14.7 | 0.538 |
| 1100 | 13.7 | 0.597 |
| 1050 | 13.0 | 0.655 |
| 1000 | 12.5 | 0.710 |
| 950 | 12.2 | 0.766 |
| 900 | 11.9 | 0.820 |
| 850 | 11.7 | 0.876 |
| 800 | 11.6 | 0.931 |
| 750 | 11.5 | 0.988 |
| 700 | 11.5 | 1.047 |
| 650 | 11.6 | 1.107 |
| 600 | 11.8 | 1.171 |
| 550 | 12.0 | 1.237 |
| 500 | 12.3 | 1.308 |
| 450 | 12.8 | 1.384 |
| 400 | 13.3 | 1.467 |
| 350 | 14.0 | 1.559 |
| 300 | 15.0 | 1.663 |
| 250 | 16.4 | 1.784 |
| 200 | 18.3 | 1.929 |
| 150 | 21.4 | 2.114 |
| 100 | 27.0 | 2.373 |




***Discussion*** The shape is not actually to scale since the horizontal axis is pressure rather than distance. If the pressure decreases linearly with distance, then the shape *would* be to scale.


**12-154** 

**Solution** Using the compressible flow relations, the one-dimensional compressible flow functions are to be evaluated and tabulated as in Table A-13 for air.

***Properties*** The specific heat ratio is given to be $k$ = 1.4 for air.

***Analysis*** The compressible flow functions listed below are expressed in EES and the results are tabulated.

$$\mathrm{Ma}^* = \mathrm{Ma}\sqrt{\frac{k+1}{2+(k-1)\mathrm{Ma}^2}} \qquad \frac{A}{A^*} = \frac{1}{\mathrm{Ma}}\left[\left(\frac{2}{k+1}\right)\left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)\right]^{0.5(k+1)/(k-1)}$$

$$\frac{P}{P_0} = \left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)^{-k/(k-1)} \qquad \frac{\rho}{\rho_0} = \left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)^{-1/(k-1)}$$

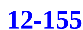
$$\frac{T}{T_0} = \left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)^{-1}$$

**Air:**

```
k=1.4
PP0=(1+(k-1)*M^2/2)^(-k/(k-1))
TT0=1/(1+(k-1)*M^2/2)
DD0=(1+(k-1)*M^2/2)^(-1/(k-1))
Mcr=M*SQRT((k+1)/(2+(k-1)*M^2))
AAcr=((2/(k+1))*(1+0.5*(k-1)*M^2))^(0.5*(k+1)/(k-1))/M
```

| Ma | Ma* | $A/A^*$ | $P/P_0$ | $\rho/\rho_0$ | $T/T_0$ |
|---|---|---|---|---|---|
| 1.0 | 1.0000 | 1.0000 | 0.5283 | 0.6339 | 0.8333 |
| 1.5 | 1.3646 | 1.1762 | 0.2724 | 0.3950 | 0.6897 |
| 2.0 | 1.6330 | 1.6875 | 0.1278 | 0.2300 | 0.5556 |
| 2.5 | 1.8257 | 2.6367 | 0.0585 | 0.1317 | 0.4444 |
| 3.0 | 1.9640 | 4.2346 | 0.0272 | 0.0762 | 0.3571 |
| 3.5 | 2.0642 | 6.7896 | 0.0131 | 0.0452 | 0.2899 |
| 4.0 | 2.1381 | 10.7188 | 0.0066 | 0.0277 | 0.2381 |
| 4.5 | 2.1936 | 16.5622 | 0.0035 | 0.0174 | 0.1980 |
| 5.0 | 2.2361 | 25.0000 | 0.0019 | 0.0113 | 0.1667 |
| 5.5 | 2.2691 | 36.8690 | 0.0011 | 0.0076 | 0.1418 |
| 6.0 | 2.2953 | 53.1798 | 0.0006 | 0.0052 | 0.1220 |
| 6.5 | 2.3163 | 75.1343 | 0.0004 | 0.0036 | 0.1058 |
| 7.0 | 2.3333 | 104.1429 | 0.0002 | 0.0026 | 0.0926 |
| 7.5 | 2.3474 | 141.8415 | 0.0002 | 0.0019 | 0.0816 |
| 8.0 | 2.3591 | 190.1094 | 0.0001 | 0.0014 | 0.0725 |
| 8.5 | 2.3689 | 251.0862 | 0.0001 | 0.0011 | 0.0647 |
| 9.0 | 2.3772 | 327.1893 | 0.0000 | 0.0008 | 0.0581 |
| 9.5 | 2.3843 | 421.1314 | 0.0000 | 0.0006 | 0.0525 |
| 10.0 | 2.3905 | 535.9375 | 0.0000 | 0.0005 | 0.0476 |

***Discussion*** The tabulated values are useful for quick calculations, but be careful – they apply only to one specific value of $k$, in this case $k$ = 1.4.


**12-155** 

**Solution** Using the compressible flow relations, the one-dimensional compressible flow functions are to be evaluated and tabulated as in Table A-13 for methane.

***Properties*** The specific heat ratio is given to be $k$ = 1.3 for methane.

***Analysis*** The compressible flow functions listed below are expressed in EES and the results are tabulated.

$$\mathrm{Ma}^* = \mathrm{Ma}\sqrt{\frac{k+1}{2+(k-1)\mathrm{Ma}^2}} \qquad \frac{A}{A^*} = \frac{1}{\mathrm{Ma}}\left[\left(\frac{2}{k+1}\right)\left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)\right]^{0.5(k+1)/(k-1)}$$

$$\frac{P}{P_0} = \left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)^{-k/(k-1)} \qquad \frac{\rho}{\rho_0} = \left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)^{-1/(k-1)}$$

$$\frac{T}{T_0} = \left(1+\frac{k-1}{2}\mathrm{Ma}^2\right)^{-1}$$

**Methane:**

```
k=1.3
PP0=(1+(k-1)*M^2/2)^(-k/(k-1))
TT0=1/(1+(k-1)*M^2/2)
DD0=(1+(k-1)*M^2/2)^(-1/(k-1))
Mcr=M*SQRT((k+1)/(2+(k-1)*M^2))
AAcr=((2/(k+1))*(1+0.5*(k-1)*M^2))^(0.5*(k+1)/(k-1))/M
```

| Ma | Ma$^*$ | $A/A^*$ | $P/P_0$ | $\rho/\rho_0$ | $T/T_0$ |
|---|---|---|---|---|---|
| 1.0 | 1.0000 | 1.0000 | 0.5457 | 0.6276 | 0.8696 |
| 1.5 | 1.3909 | 1.1895 | 0.2836 | 0.3793 | 0.7477 |
| 2.0 | 1.6956 | 1.7732 | 0.1305 | 0.2087 | 0.6250 |
| 2.5 | 1.9261 | 2.9545 | 0.0569 | 0.1103 | 0.5161 |
| 3.0 | 2.0986 | 5.1598 | 0.0247 | 0.0580 | 0.4255 |
| 3.5 | 2.2282 | 9.1098 | 0.0109 | 0.0309 | 0.3524 |
| 4.0 | 2.3263 | 15.9441 | 0.0050 | 0.0169 | 0.2941 |
| 4.5 | 2.4016 | 27.3870 | 0.0024 | 0.0095 | 0.2477 |
| 5.0 | 2.4602 | 45.9565 | 0.0012 | 0.0056 | 0.2105 |
| 5.5 | 2.5064 | 75.2197 | 0.0006 | 0.0033 | 0.1806 |
| 6.0 | 2.5434 | 120.0965 | 0.0003 | 0.0021 | 0.1563 |
| 6.5 | 2.5733 | 187.2173 | 0.0002 | 0.0013 | 0.1363 |
| 7.0 | 2.5978 | 285.3372 | 0.0001 | 0.0008 | 0.1198 |
| 7.5 | 2.6181 | 425.8095 | 0.0001 | 0.0006 | 0.1060 |
| 8.0 | 2.6350 | 623.1235 | 0.0000 | 0.0004 | 0.0943 |
| 8.5 | 2.6493 | 895.5077 | 0.0000 | 0.0003 | 0.0845 |
| 9.0 | 2.6615 | 1265.6040 | 0.0000 | 0.0002 | 0.0760 |
| 9.5 | 2.6719 | 1761.2133 | 0.0000 | 0.0001 | 0.0688 |
| 10.0 | 2.6810 | 2416.1184 | 0.0000 | 0.0001 | 0.0625 |

***Discussion*** The tabulated values are useful for quick calculations, but be careful – they apply only to one specific value of $k$, in this case $k$ = 1.3.



**12-156** 

**Solution** Using the normal shock relations, the normal shock functions are to be evaluated and tabulated as in Table A-14 for air.

***Properties*** The specific heat ratio is given to be $k$ = 1.4 for air.

***Analysis*** The normal shock relations listed below are expressed in EES and the results are tabulated.

$$\mathrm{Ma}_2 = \sqrt{\frac{(k-1)\mathrm{Ma}_1^2 + 2}{2k\mathrm{Ma}_1^2 - k + 1}} \qquad \frac{P_2}{P_1} = \frac{1 + k\mathrm{Ma}_1^2}{1 + k\mathrm{Ma}_2^2} = \frac{2k\mathrm{Ma}_1^2 - k + 1}{k+1}$$

$$\frac{T_2}{T_1} = \frac{2 + \mathrm{Ma}_1^2(k-1)}{2 + \mathrm{Ma}_2^2(k-1)} \qquad \frac{\rho_2}{\rho_1} = \frac{P_2/P_1}{T_2/T_1} = \frac{(k+1)\mathrm{Ma}_1^2}{2 + (k-1)\mathrm{Ma}_1^2} = \frac{V_1}{V_2},$$

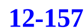
$$\frac{P_{02}}{P_{01}} = \frac{\mathrm{Ma}_1}{\mathrm{Ma}_2}\left[\frac{1 + \mathrm{Ma}_2^2(k-1)/2}{1 + \mathrm{Ma}_1^2(k-1)/2}\right]^{\frac{k+1}{2(k-1)}} \qquad \frac{P_{02}}{P_1} = \frac{(1 + k\mathrm{Ma}_1^2)[1 + \mathrm{Ma}_2^2(k-1)/2]^{k/(k-1)}}{1 + k\mathrm{Ma}_2^2}$$

**Air:**

```
k=1.4
My=SQRT((Mx^2+2/(k-1))/(2*Mx^2*k/(k-1)-1))
PyPx=(1+k*Mx^2)/(1+k*My^2)
TyTx=(1+Mx^2*(k-1)/2)/(1+My^2*(k-1)/2)
RyRx=PyPx/TyTx
P0yP0x=(Mx/My)*((1+My^2*(k-1)/2)/(1+Mx^2*(k-1)/2))^(0.5*(k+1)/(k-1))
P0yPx=(1+k*Mx^2)*(1+My^2*(k-1)/2)^(k/(k-1))/(1+k*My^2)
```

| $\mathrm{Ma}_1$ | $\mathrm{Ma}_2$ | $P_2/P_1$ | $\rho_2/\rho_1$ | $T_2/T_1$ | $P_{02}/P_{01}$ | $P_{02}/P_1$ |
|---|---|---|---|---|---|---|
| 1.0 | 1.0000 | 1.0000 | 1.0000 | 1.0000 | 1 | 1.8929 |
| 1.5 | 0.7011 | 2.4583 | 1.8621 | 1.3202 | 0.9298 | 3.4133 |
| 2.0 | 0.5774 | 4.5000 | 2.6667 | 1.6875 | 0.7209 | 5.6404 |
| 2.5 | 0.5130 | 7.1250 | 3.3333 | 2.1375 | 0.499 | 8.5261 |
| 3.0 | 0.4752 | 10.3333 | 3.8571 | 2.6790 | 0.3283 | 12.0610 |
| 3.5 | 0.4512 | 14.1250 | 4.2609 | 3.3151 | 0.2129 | 16.2420 |
| 4.0 | 0.4350 | 18.5000 | 4.5714 | 4.0469 | 0.1388 | 21.0681 |
| 4.5 | 0.4236 | 23.4583 | 4.8119 | 4.8751 | 0.0917 | 26.5387 |
| 5.0 | 0.4152 | 29.0000 | 5.0000 | 5.8000 | 0.06172 | 32.6535 |
| 5.5 | 0.4090 | 35.1250 | 5.1489 | 6.8218 | 0.04236 | 39.4124 |
| 6.0 | 0.4042 | 41.8333 | 5.2683 | 7.9406 | 0.02965 | 46.8152 |
| 6.5 | 0.4004 | 49.1250 | 5.3651 | 9.1564 | 0.02115 | 54.8620 |
| 7.0 | 0.3974 | 57.0000 | 5.4444 | 10.4694 | 0.01535 | 63.5526 |
| 7.5 | 0.3949 | 65.4583 | 5.5102 | 11.8795 | 0.01133 | 72.8871 |
| 8.0 | 0.3929 | 74.5000 | 5.5652 | 13.3867 | 0.008488 | 82.8655 |
| 8.5 | 0.3912 | 84.1250 | 5.6117 | 14.9911 | 0.006449 | 93.4876 |
| 9.0 | 0.3898 | 94.3333 | 5.6512 | 16.6927 | 0.004964 | 104.7536 |
| 9.5 | 0.3886 | 105.1250 | 5.6850 | 18.4915 | 0.003866 | 116.6634 |
| 10.0 | 0.3876 | 116.5000 | 5.7143 | 20.3875 | 0.003045 | 129.2170 |

***Discussion*** The tabulated values are useful for quick calculations, but be careful – they apply only to one specific value of $k$, in this case $k$ = 1.4.


**12-157** 

**Solution** Using the normal shock relations, the normal shock functions are to be evaluated and tabulated as in Table A-14 for methane.

***Properties*** The specific heat ratio is given to be $k$ = 1.3 for methane.

***Analysis*** The normal shock relations listed below are expressed in EES and the results are tabulated.

$$\text{Ma}_2 = \sqrt{\frac{(k-1)\text{Ma}_1^2 + 2}{2k\text{Ma}_1^2 - k + 1}} \qquad \frac{P_2}{P_1} = \frac{1 + k\text{Ma}_1^2}{1 + k\text{Ma}_2^2} = \frac{2k\text{Ma}_1^2 - k + 1}{k+1}$$

$$\frac{T_2}{T_1} = \frac{2 + \text{Ma}_1^2(k-1)}{2 + \text{Ma}_2^2(k-1)} \qquad \frac{\rho_2}{\rho_1} = \frac{P_2/P_1}{T_2/T_1} = \frac{(k+1)\text{Ma}_1^2}{2 + (k-1)\text{Ma}_1^2} = \frac{V_1}{V_2},$$

$$\frac{P_{02}}{P_{01}} = \frac{\text{Ma}_1}{\text{Ma}_2}\left[\frac{1 + \text{Ma}_2^2(k-1)/2}{1 + \text{Ma}_1^2(k-1)/2}\right]^{\frac{k+1}{2(k-1)}} \qquad \frac{P_{02}}{P_1} = \frac{(1 + k\text{Ma}_1^2)[1 + \text{Ma}_2^2(k-1)/2]^{k/(k-1)}}{1 + k\text{Ma}_2^2}$$

**Methane:**

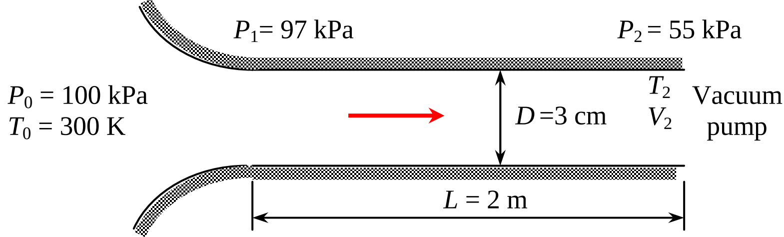
```
k=1.3
My=SQRT((Mx^2+2/(k-1))/(2*Mx^2*k/(k-1)-1))
PyPx=(1+k*Mx^2)/(1+k*My^2)
TyTx=(1+Mx^2*(k-1)/2)/(1+My^2*(k-1)/2)
RyRx=PyPx/TyTx
P0yP0x=(Mx/My)*((1+My^2*(k-1)/2)/(1+Mx^2*(k-1)/2))^(0.5*(k+1)/(k-1))
P0yPx=(1+k*Mx^2)*(1+My^2*(k-1)/2)^(k/(k-1))/(1+k*My^2)
```

| $\text{Ma}_1$ | $\text{Ma}_2$ | $P_2/P_1$ | $\rho_2/\rho_1$ | $T_2/T_1$ | $P_{02}/P_{01}$ | $P_{02}/P_1$ |
|---|---|---|---|---|---|---|
| 1.0 | 1.0000 | 1.0000 | 1.0000 | 1.0000 | 1 | 1.8324 |
| 1.5 | 0.6942 | 2.4130 | 1.9346 | 1.2473 | 0.9261 | 3.2654 |
| 2.0 | 0.5629 | 4.3913 | 2.8750 | 1.5274 | 0.7006 | 5.3700 |
| 2.5 | 0.4929 | 6.9348 | 3.7097 | 1.8694 | 0.461 | 8.0983 |
| 3.0 | 0.4511 | 10.0435 | 4.4043 | 2.2804 | 0.2822 | 11.4409 |
| 3.5 | 0.4241 | 13.7174 | 4.9648 | 2.7630 | 0.1677 | 15.3948 |
| 4.0 | 0.4058 | 17.9565 | 5.4118 | 3.3181 | 0.09933 | 19.9589 |
| 4.5 | 0.3927 | 22.7609 | 5.7678 | 3.9462 | 0.05939 | 25.1325 |
| 5.0 | 0.3832 | 28.1304 | 6.0526 | 4.6476 | 0.03613 | 30.9155 |
| 5.5 | 0.3760 | 34.0652 | 6.2822 | 5.4225 | 0.02243 | 37.3076 |
| 6.0 | 0.3704 | 40.5652 | 6.4688 | 6.2710 | 0.01422 | 44.3087 |
| 6.5 | 0.3660 | 47.6304 | 6.6218 | 7.1930 | 0.009218 | 51.9188 |
| 7.0 | 0.3625 | 55.2609 | 6.7485 | 8.1886 | 0.006098 | 60.1379 |
| 7.5 | 0.3596 | 63.4565 | 6.8543 | 9.2579 | 0.004114 | 68.9658 |
| 8.0 | 0.3573 | 72.2174 | 6.9434 | 10.4009 | 0.002827 | 78.4027 |
| 8.5 | 0.3553 | 81.5435 | 7.0190 | 11.6175 | 0.001977 | 88.4485 |
| 9.0 | 0.3536 | 91.4348 | 7.0837 | 12.9079 | 0.001404 | 99.1032 |
| 9.5 | 0.3522 | 101.8913 | 7.1393 | 14.2719 | 0.001012 | 110.367 |
| 10.0 | 0.3510 | 112.9130 | 7.1875 | 15.7096 | 0.000740 | 122.239 |

***Discussion*** The tabulated values are useful for quick calculations, but be careful – they apply only to one specific value of $k$, in this case $k$ = 1.3.

**12-158**

**Solution** Air enters a constant-area adiabatic duct at a specified state, and leaves at a specified pressure. The mass flow rate of air, the exit velocity, and the average friction factor are to be determined.




***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor is constant along the duct.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K. The friction factor is given to be $f$ = 0.025.

***Analysis*** Noting that the flow in the nozzle section is isentropic, the Mach number, thermodynamic temperature, and density at the tube inlet become

$$P_1 = P_{01}\left(1+\frac{k-1}{2}\text{Ma}_1^2\right)^{-k/(k-1)} \rightarrow \quad 97 \text{ kPa} = (100 \text{ kPa})\left(1+\frac{1.4-1}{2}\text{Ma}_1^2\right)^{-1.4/0.4} \rightarrow \quad \text{Ma}_1 = 0.2091$$

$$T_1 = T_{01}\left(1+\frac{k-1}{2}\text{Ma}_1^2\right)^{-1} = (300\text{ K})\left(1+\frac{1.4-1}{2}(0.2091)^2\right)^{-1} = 297.4 \text{ K}$$

$$\rho_1 = \frac{P_1}{RT_1} = \frac{97 \text{ kPa}}{(0.287 \text{ kJ/kgK})(297.4 \text{ K})} = 1.136 \text{ kg/m}^3$$

Then the inlet velocity and the mass flow rate become

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287 \text{ kJ/kg}\cdot\text{K})(297.4 \text{ K})\left(\frac{1000 \text{ m}^2/\text{s}^2}{1 \text{ kJ/kg}}\right)} = 345.7 \text{ m/s}$$

$$V_1 = \text{Ma}_1 c_1 = 0.2091(345.7 \text{ m/s}) = 72.3 \text{ m/s}$$

$$\dot{m}_{air} = \rho_1 A_{c1} V_1 = (1.136 \text{ kg/m}^3)[\pi(0.03\text{ m})^2/4](72.3 \text{ m/s}) = \mathbf{0.0581 \text{ kg/s}}$$

The Fanno flow functions corresponding to the inlet Mach number are, from Table A-16,

$\text{Ma}_1 = 0.2091$: $(fL^*/D_h)_1$ =13.095 $T_1/T^*$ = 1.1896, $P_1/P^*$ = 5.2173, $V_1/V^*$ = 0.2280

Therefore, $P_1 = 5.2173P^*$. Then the Fanno function $P_2/P^*$ becomes

$$\frac{P_2}{P^*} = \frac{P_2}{P_1/5.2173} = \frac{5.2173(55 \text{ kPa})}{97 \text{ kPa}} = 2.9583$$

The corresponding Mach number and Fanno flow functions are, from Table A-16,

$\text{Ma}_2 = 0.3655$, $(fL^*/D_h)_1$ = 3.0420, and $V_2/V^*$ =0.3951.

Then the air velocity at the duct exit and the average friction factor become

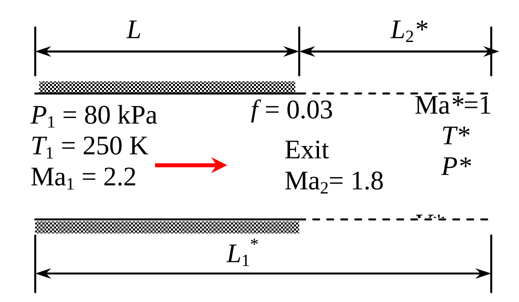
$$\frac{V_2}{V_1} = \frac{V_2/V^*}{V_1/V^*} = \frac{0.3951}{0.2280} = 1.7329 \qquad \rightarrow V_2 = 1.7329V_1 = 1.7329(72.3 \text{ m/s}) = \mathbf{125 \text{ m/s}}$$

$$L = L_1^* - L_2^* = \left(\frac{fL_1^*}{D_h} - \frac{fL_2^*}{D_h}\right)\frac{D_h}{f} \rightarrow \quad 2 \text{ m} = (13.095 - 3.042)\frac{0.03\text{ m}}{f} \quad \rightarrow f = \mathbf{0.151}$$

***Discussion*** Note that the mass flow rate and the average friction factor can be determined by measuring static pressure, as in incompressible flow.

**12-159**

**Solution** Supersonic airflow in a constant cross-sectional area adiabatic duct is considered. For a specified exit Mach number, the temperature, pressure, and velocity at the duct exit are to be determined.




***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor remains constant along the duct.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K. The average friction factor is given to be $f$ = 0.03.

***Analysis*** The inlet velocity is

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(250\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 316.9\text{ m/s}$$

$$V_1 = \text{Ma}_1 c_1 = 2.2(316.9\text{ m/s}) = 697.3\text{ m/s}$$

The Fanno flow functions corresponding to the inlet and exit Mach numbers are, from Table A-16,

$\text{Ma}_1 = 2.2$: $(fL^*/D_h)_1 = 0.3609$ $T_1/T^* = 0.6098$, $P_1/P^* = 0.3549$, $V_1/V^* = 1.7179$

$\text{Ma}_2 = 1.8$: $(fL^*/D_h)_2 = 0.2419$ $T_2/T^* = 0.7282$, $P_2/P^* = 0.4741$, $V_2/V^* = 1.5360$

Then the temperature, pressure, and velocity at the duct exit are determined to be

$$\frac{T_2}{T_1} = \frac{T_2/T^*}{T_1/T^*} = \frac{0.7282}{0.6098} = 1.1942 \quad \rightarrow \quad T_2 = 1.1942T_1 = 1.1942(250\text{ K}) = \mathbf{299\ K}$$

$$\frac{P_2}{P_1} = \frac{P_2/P^*}{P_1/P^*} = \frac{0.4741}{0.3549} = 1.3359 \quad \rightarrow \quad P_2 = 1.3359P_1 = 1.3359(80\text{ kPa}) = \mathbf{107\ kPa}$$

$$\frac{V_2}{V_1} = \frac{V_2/V^*}{V_1/V^*} = \frac{1.5360}{1.7179} = 0.8941 \quad \rightarrow \quad V_2 = 0.8941V_1 = 0.8941(697.3\text{ m/s}) = \mathbf{623\ m/s}$$

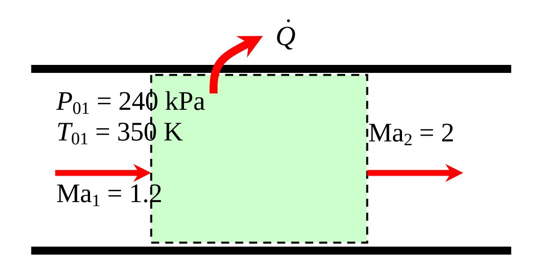
***Discussion*** The duct length is determined to be

$$L = L_1^* - L_2^* = \left(\frac{fL_1^*}{D_h} - \frac{fL_2^*}{D_h}\right)\frac{D_h}{f} = (0.3609 - 0.2419)\frac{0.04\text{ m}}{0.03} = \mathbf{0.16\ m}$$

Note that it takes a duct length of only 0.16 m for the Mach number to decrease from 2.2 to 1.8. The maximum (or sonic) duct lengths at the inlet and exit states in this case are $L_1^*$ = 0.48 m and $L_2^*$ = 0.32 m. Therefore, the flow would reach sonic conditions if a 0.32-m long section were added to the existing duct.

**12-160**

**Solution** Air flowing at a supersonic velocity in a duct is accelerated by cooling. For a specified exit Mach number, the rate of heat transfer is to be determined.




***Assumptions*** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K.

***Analysis*** Knowing stagnation properties, the static properties are determined to be

$$T_1 = T_{01}\left(1+\frac{k-1}{2}\mathrm{Ma}_1^2\right)^{-1} = (350\ \mathrm{K})\left(1+\frac{1.4\text{-}1}{2}1.2^2\right)^{-1} = 271.7\ \mathrm{K}$$

$$P_1 = P_{01}\left(1+\frac{k-1}{2}\mathrm{Ma}_1^2\right)^{-k/(k-1)} = (240\ \mathrm{kPa})\left(1+\frac{1.4\text{-}1}{2}1.2^2\right)^{-1.4/0.4} = 98.97\ \mathrm{kPa}$$

$$\rho_1 = \frac{P_1}{RT_1} = \frac{98.97\ \mathrm{kPa}}{(0.287\ \mathrm{kJ/kgK})(271.7\ \mathrm{K})} = 1.269\ \mathrm{kg/m^3}$$

Then the inlet velocity and the mass flow rate become

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\ \mathrm{kJ/kg\cdot K})(271.7\ \mathrm{K})\left(\frac{1000\ \mathrm{m^2/s^2}}{1\ \mathrm{kJ/kg}}\right)} = 330.4\ \mathrm{m/s}$$

$$V_1 = \mathrm{Ma}_1 c_1 = 1.2(330.4\ \mathrm{m/s}) = 396.5\ \mathrm{m/s}$$

$$\dot{m}_{air} = \rho_1 A_{c1} V_1 = (1.269\ \mathrm{kg/m^3})[\pi(0.20\ \mathrm{m})^2/4](396.5\ \mathrm{m/s}) = 15.81\ \mathrm{kg/s}$$

The Rayleigh flow functions $T_0/T_0^*$ corresponding to the inlet and exit Mach numbers are (Table A-15):

$\mathrm{Ma}_1 = 1.8$: $T_{01}/T_0^* = 0.9787$
$\mathrm{Ma}_2 = 2$: $T_{02}/T_0^* = 0.7934$

Then the exit stagnation temperature is determined to be

$$\frac{T_{02}}{T_{01}} = \frac{T_{02}/T_0^*}{T_{01}/T_0^*} = \frac{0.7934}{0.9787} = 0.8107 \quad \rightarrow \quad T_{02} = 0.8107T_{01} = 0.8107(350\ \mathrm{K}) = 283.7\ \mathrm{K}$$

Finally, the rate of heat transfer is

$$\dot{Q} = \dot{m}_{\mathrm{air}} c_p \left(T_{02} - T_{01}\right) = (15.81\ \mathrm{kg/s})(1.005\ \mathrm{kJ/kg\cdot K})(283.7-350)\ \mathrm{K} = -1053\ \mathrm{kW} \cong \mathbf{-1050\ kW}$$

***Discussion*** The negative sign confirms that the gas needs to be cooled in order to be accelerated. Also, it can be shown that the thermodynamic temperature drops to 158 K at the exit, which is extremely low. Therefore, the duct may need to be heavily insulated to maintain indicated flow conditions.

**12-161**

**Solution** Air flowing at a subsonic velocity in a duct is accelerated by heating. The highest rate of heat transfer without affecting the inlet conditions is to be determined.

***Assumptions*** **1** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid. **2** Inlet conditions (and thus the mass flow rate) remain constant.

***Properties*** We take the properties of air to be $k = 1.4$, $c_p = 1.005$ kJ/kg·K, and $R = 0.287$ kJ/kg·K.

***Analysis*** Heat transfer will stop when the flow is choked, and thus $Ma_2 = V_2/c_2 = 1$. The inlet density and stagnation temperature are

$$\rho_1 = \frac{P_1}{RT_1} = \frac{400\text{ kPa}}{(0.287\text{ kJ/kgK})(360\text{ K})} = 3.871\text{ kg/m}^3$$

$$T_{01} = T_1\left(1+\frac{k-1}{2}\text{Ma}_1^2\right) = (360\text{ K})\left(1+\frac{1.4\text{-}1}{2}0.4^2\right) = 371.5\text{ K}$$

$\dot{Q}$

$P_1 = 400$ kPa

$T_1 = 360$ K

$Ma_1 = 0.4$

$Ma_2 = 1$

Then the inlet velocity and the mass flow rate become

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(360\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 380.3\text{ m/s}$$

$$V_1 = \text{Ma}_1 c_1 = 0.4(380.3\text{ m/s}) = 152.1\text{ m/s}$$

$$\dot{m}_{air} = \rho_1 A_{c1} V_1 = (3.871\text{ kg/m}^3)(0.1\times 0.1\text{ m}^2)(152.1\text{ m/s}) = 5.890\text{ kg/s}$$

The Rayleigh flow functions corresponding to the inlet and exit Mach numbers are $T_{02}/T_0^* = 1$ (since $Ma_2 = 1$) .

$$\frac{T_{01}}{T_0^*} = \frac{(k+1)\text{Ma}_1^2[2+(k-1)\text{Ma}_1^2]}{(1+k\text{Ma}_1^2)^2} = \frac{(1.4+1)0.4^2[2+(1.4-1)0.4^2]}{(1+1.4\times 0.4^2)^2} = 0.5290$$

Therefore,

$$\frac{T_{02}}{T_{01}} = \frac{T_{02}/T_0^*}{T_{01}/T_0^*} = \frac{1}{0.5290} \quad \rightarrow \quad T_{02} = T_{01}/0.5290 = (371.5\text{ K})/0.5290 = 702.3\text{ K}$$

Then the rate of heat transfer becomes

$$\dot{Q} = \dot{m}_{\text{air}} c_p \left(T_{02} - T_{01}\right) = \left(5.890\text{ kg/s}\right)\left(1.005\text{ kJ/kg}\cdot\text{K}\right)\left(702.3 - 371.5\right)\text{ K} = 1958\text{ kW} \cong \mathbf{1960\text{ kW}}$$

***Discussion*** It can also be shown that $T_2 = 585$ K, which is the highest thermodynamic temperature that can be attained under stated conditions. If more heat is transferred, the additional temperature rise will cause the mass flow rate to decrease. We can also solve this problem using the Rayleigh function values listed in Table A-15.

**12-162**

**Solution** Helium flowing at a subsonic velocity in a duct is accelerated by heating. The highest rate of heat transfer without affecting the inlet conditions is to be determined.

***Assumptions*** **1** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid. **2** Inlet conditions (and thus the mass flow rate) remain constant.

***Properties*** We take the properties of helium to be $k$ = 1.667, $c_p$ = 5.193 kJ/kg·K, and $R$ = 2.077 kJ/kg·K.

***Analysis*** Heat transfer will stop when the flow is choked, and thus $Ma_2 = V_2/c_2 = 1$. The inlet density and stagnation temperature are

$\dot{Q}$

$P_1$ = 400 kPa

$T_1$ = 360 K

$Ma_1$ = 0.4

$Ma_2$ = 1

$$\rho_1 = \frac{P_1}{RT_1} = \frac{400\text{ kPa}}{(2.077\text{ kJ/kgK})(360\text{ K})} = 0.5350\text{ kg/m}^3$$

$$T_{01} = T_1\left(1+\frac{k-1}{2}\text{Ma}_1^2\right) = (360\text{ K})\left(1+\frac{1.667-1}{2}0.4^2\right) = 379.2\text{ K}$$

Then the inlet velocity and the mass flow rate become

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.667)(2.077\text{ kJ/kg}\cdot\text{K})(360\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 1116\text{ m/s}$$

$$V_1 = \text{Ma}_1 c_1 = 0.4(1116\text{ m/s}) = 446.6\text{ m/s}$$

$$\dot{m}_{air} = \rho_1 A_{c1} V_1 = (0.5350\text{ kg/m}^3)(0.1\times 0.1\text{ m}^2)(446.6\text{ m/s}) = 2.389\text{ kg/s}$$

The Rayleigh flow functions corresponding to the inlet and exit Mach numbers are $T_{02}/T_0^* = 1$ (since $Ma_2 = 1$)

$$\frac{T_{01}}{T_0^*} = \frac{(k+1)\text{Ma}_1^2[2+(k-1)\text{Ma}_1^2]}{(1+k\text{Ma}_1^2)^2} = \frac{(1.667+1)0.4^2[2+(1.667-1)0.4^2]}{(1+1.667\times 0.4^2)^2} = 0.5603$$

Therefore,

$$\frac{T_{02}}{T_{01}} = \frac{T_{02}/T_0^*}{T_{01}/T_0^*} = \frac{1}{0.5603} \quad \rightarrow \quad T_{02} = T_{01}/0.5603 = (379.2\text{ K})/0.5603 = 676.8\text{ K}$$

Then the rate of heat transfer becomes

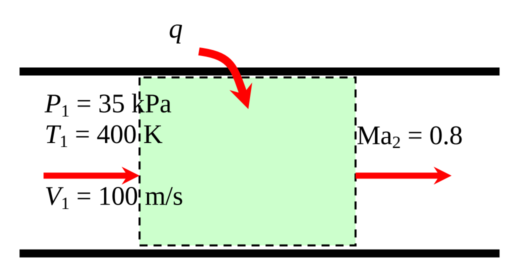
$$\dot{Q} = \dot{m}_{\text{air}} c_p (T_{02} - T_{01}) = (2.389\text{ kg/s})(5.193\text{ kJ/kg}\cdot\text{K})(676.8 - 379.2)\text{ K} = \mathbf{3690\ kW}$$

***Discussion*** It can also be shown that $T_2$ = 508 K, which is the highest thermodynamic temperature that can be attained under stated conditions. If more heat is transferred, the additional temperature rise will cause the mass flow rate to decrease. Also, in the solution of this problem, we cannot use the values of Table A-15 since they are based on $k$ = 1.4.

**12-163**

**Solution** Air flowing at a subsonic velocity in a duct is accelerated by heating. For a specified exit Mach number, the heat transfer for a specified exit Mach number as well as the maximum heat transfer are to be determined.

***Assumptions*** **1** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid. **2** Inlet conditions (and thus the mass flow rate) remain constant.




***Properties*** We take the properties of air to be $k = 1.4$, $c_p = 1.005$ kJ/kg·K, and $R = 0.287$ kJ/kg·K.

***Analysis*** The inlet Mach number and stagnation temperature are

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(400\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 400.9\text{ m/s}$$

$$\text{Ma}_1 = \frac{V_1}{c_1} = \frac{100\text{ m/s}}{400.9\text{ m/s}} = 0.2494$$

$$T_{01} = T_1\left(1 + \frac{k-1}{2}\text{Ma}_1^2\right) = (400\text{ K})\left(1 + \frac{1.4-1}{2}0.2494^2\right) = 405.0\text{ K}$$

The Rayleigh flow functions corresponding to the inlet and exit Mach numbers are (Table A-15):

$\text{Ma}_1 = 0.2494$: $T_{01}/T^* = 0.2559$
$\text{Ma}_2 = 0.8$: $T_{02}/T^* = 0.9639$

Then the exit stagnation temperature and the heat transfer are determined to be

$$\frac{T_{02}}{T_{01}} = \frac{T_{02}/T^*}{T_{01}/T^*} = \frac{0.9639}{0.2559} = 3.7667 \quad \rightarrow \quad T_{02} = 3.7667T_{01} = 3.7667(405.0\text{ K}) = 1526\text{ K}$$

$$q = c_p\left(T_{02} - T_{01}\right) = \left(1.005\text{ kJ/kg}\cdot\text{K}\right)\left(1526 - 405\right)\text{ K} = 1126\text{ kJ/kg} \cong \mathbf{1130\ kJ/kg}$$

Maximum heat transfer will occur when the flow is choked, and thus $\text{Ma}_2 = 1$ and thus $T_{02}/T^* = 1$. Then,

$$\frac{T_{02}}{T_{01}} = \frac{T_{02}/T^*}{T_{01}/T^*} = \frac{1}{0.2559} \quad \rightarrow \quad T_{02} = T_{01}/0.2559 = (405\text{ K})/0.2559 = 1583\text{ K}$$

$$q_{max} = c_p(T_{02} - T_{01}) = (1.005\text{ kJ/kg}\cdot\text{K})(1583 - 405)\text{ K} = 1184\text{ kJ/kg} \cong \mathbf{1180\ kJ/kg}$$

***Discussion*** This is the maximum heat that can be transferred to the gas without affecting the mass flow rate. If more heat is transferred, the additional temperature rise will cause the mass flow rate to decrease.

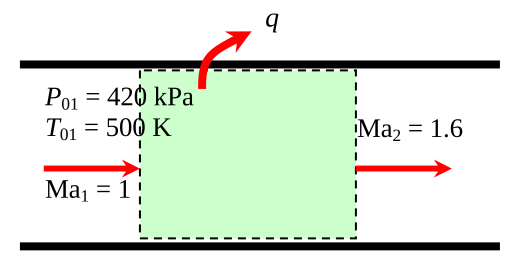
**12-164**

**Solution** Air flowing at sonic conditions in a duct is accelerated by cooling. For a specified exit Mach number, the amount of heat transfer per unit mass is to be determined.

***Assumptions*** The assumptions associated with Rayleigh flow (i.e., steady one-dimensional flow of an ideal gas with constant properties through a constant cross-sectional area duct with negligible frictional effects) are valid.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K.

***Analysis*** Noting that $Ma_1$ = 1, the inlet stagnation temperature is




$$T_{01} = T_1\left(1+\frac{k-1}{2}\text{Ma}_1^2\right) = (500\text{ K})\left(1+\frac{1.4-1}{2}1^2\right) = 600\text{ K}$$

The Rayleigh flow functions $T_0/T_0^*$ corresponding to the inlet and exit Mach numbers are (Table A-15):

$Ma_1$ = 1: $T_{01}/T_0^*$ = 1
$Ma_2$ = 1.6: $T_{02}/T_0^*$ = 0.8842

Then the exit stagnation temperature and heat transfer are determined to be

$$\frac{T_{02}}{T_{01}} = \frac{T_{02}/T_0^*}{T_{01}/T_0^*} = \frac{0.8842}{1} = 0.8842 \quad \rightarrow \quad T_{02} = 0.8842T_{01} = 0.8842(600\text{ K}) = 530.5\text{ K}$$

$$q = c_p(T_{02}-T_{01}) = (1.005\text{ kJ/kg}\cdot\text{K})(530.5-600)\text{ K} = \mathbf{-69.8\ kJ/kg}$$

***Discussion*** The negative sign confirms that the gas needs to be cooled in order to be accelerated. Also, it can be shown that the thermodynamic temperature drops to 351 K at the exit

**12-165**

**Solution** Combustion gases enter a constant-area adiabatic duct at a specified state, and undergo a normal shock at a specified location. The exit velocity, temperature, and pressure are to be determined.

***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor is constant along the duct.

***Properties*** The specific heat ratio and gas constant of combustion gases are given to be $k$ = 1.33 and $R$ = 0.280 kJ/kg·K. The friction factor is given to be $f$ = 0.010.

$Ma^* = 1$

$P_1$ = 180 kPa
$T_1$ = 510 K
$Ma_1$ = 2

Normal shock

$T^*$
$P^*$
$V^*$

$L_1$ = 2 m

***Analysis*** The Fanno flow functions corresponding to the inlet Mach number of 2 are calculated from the relations in Table A-16 for $k$ = 1.33 to be

$Ma_1 = 2$: $(fL^*/D_h)_1 = 0.3402$ $T_1/T^* = 0.7018$, $P_1/P^* = 0.4189$

First we check to make sure that the flow everywhere upstream the shock is supersonic. The required duct length from the inlet $L_1^*$ for the flow to reach sonic conditions is $L_1^* = 0.3402\dfrac{D}{f} = 0.3402\dfrac{0.10\text{ m}}{0.010} = 3.40\text{ m}$, which is greater than the actual length of 2 m. Therefore, the flow is indeed supersonic when the normal shock occurs at the indicated location. Also, using the actual duct length $L_1$, we have $\dfrac{fL_1}{D_h} = \dfrac{(0.010)(2\text{ m})}{0.10\text{ m}} = 0.2000$. Noting that $L_1 = L_1^* - L_2^*$, the function $fL^*/D_h$ at the exit state and the corresponding Mach number are $\left(\dfrac{fL^*}{D_h}\right)_2 = \left(\dfrac{fL^*}{D_h}\right)_1 - \dfrac{fL_1}{D_h} = 0.3402 - 0.2000 = 0.1402 \rightarrow Ma_2 = 1.476$.

From the relations in Table A-16, at $Ma_2$ =1.476: $T_2/T^* = 0.8568$, $P_2/P^* = 0.6270$. Then the temperature, pressure, and velocity before the shock are determined to be

$$\frac{T_2}{T_1} = \frac{T_2/T^*}{T_1/T^*} = \frac{0.8568}{0.7018} = 1.2209 \quad \rightarrow \quad T_2 = 1.2209T_1 = 1.2209(510\text{ K}) = 622.7\text{ K}$$

$$\frac{P_2}{P_1} = \frac{P_2/P^*}{P_1/P^*} = \frac{0.6270}{0.4189} = 1.4968 \quad \rightarrow \quad P_2 = 1.4968P_1 = 1.4968(180\text{ kPa}) = 269.4\text{ kPa}$$

The normal shock functions corresponding to a Mach number of 1.476 are, from the relations in Table A-14,

$Ma_2$ =1.476: $Ma_3 = 0.7052$, $T_3/T_2 = 1.2565$, $P_3/P_2 = 2.3466$

Then the temperature and pressure after the shock become

$$T_3 = 1.2565T_2 = 1.2565(622.7\text{ K}) = 782.4\text{ K}$$

$$P_3 = 2.3466P_2 = 2.3466(269.4\text{ kPa}) = 632.3\text{ kPa}$$

Sonic conditions exist at the duct exit, and the flow downstream of the shock is still Fanno flow. From the relations in Table A-16,

$Ma_3 = 0.7052$: $T_3/T^* = 1.0767$, $P_3/P^* = 1.4713$

$Ma_4 = 1$: $T_4/T^* = 1$, $P_4/P^* = 1$

Then the temperature, pressure, and velocity at the duct exit are determined to be

$$\frac{T_4}{T_3} = \frac{T_4/T^*}{T_3/T^*} = \frac{1}{1.0767} \quad \rightarrow \quad T_4 = T_3/1.0767 = (782.4\text{ K})/1.0767 = \mathbf{727\ K}$$

$$\frac{P_4}{P_3} = \frac{P_4/P^*}{P_3/P^*} = \frac{1}{1.4713} \quad \rightarrow \quad P_4 = P_3/1.4713 = (632.3\text{ kPa})/1.4713 = \mathbf{430\ kPa}$$

$$V_4 = Ma_4 c_4 = (1)\sqrt{kRT_4} = \sqrt{(1.33)(0.280\text{ kJ/kg}\cdot\text{K})(727\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = \mathbf{520\ m/s}$$

***Discussion*** It can be shown that $L_3^*$ = 2.13 m, and thus the total length of this duct is 4.13 m. If the duct is extended, the normal shock will move farther upstream, and eventually to the inlet of the duct.



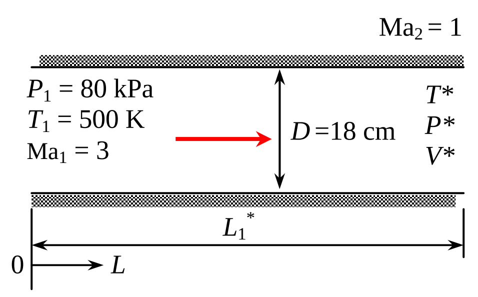

**12-166**

**Solution** Choked supersonic airflow in a constant cross-sectional area adiabatic duct is considered. The variation of duct length with Mach number is to be investigated, and the results are to be plotted.

***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor remains constant along the duct.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K. The average friction factor is given to be $f$ = 0.03.

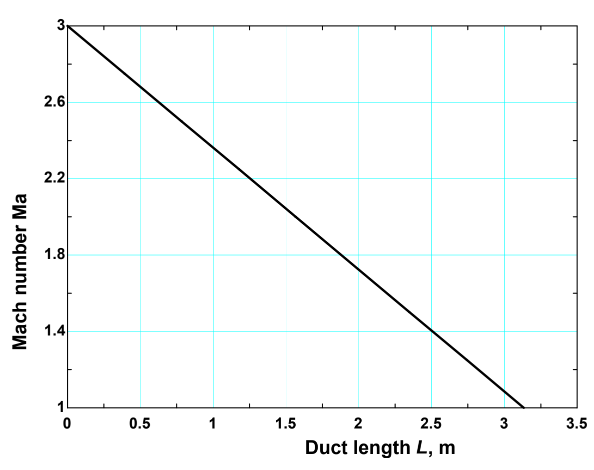
***Analysis*** We use EES to solve the problem. The flow is choked, and thus $Ma_2$ =1. Corresponding to the inlet Mach number of $Ma_1$ = 3 we have, from Table A-16, $fL^*/D_h$ = 0.5222, Therefore, the original duct length is




$$L_1^* = 0.5222\frac{D}{f} = 0.5222\frac{0.18\text{ m}}{0.03} = 3.13\text{ m}$$

Repeating the calculations for different $Ma_2$ as it varies from 3 to 1 results in the following table for the location on the duct from the inlet:

| Mach number, Ma | Duct length $L$, m |
|---|---|
| 3.00 | 0.00 |
| 2.75 | 0.39 |
| 2.50 | 0.78 |
| 2.25 | 1.17 |
| 2.00 | 1.57 |
| 1.75 | 1.96 |
| 1.50 | 2.35 |
| 1.25 | 2.74 |
| 1.00 | 3.13 |




**EES program:**

```
k=1.4
cp=1.005
R=0.287

P1=80
T1=500
Ma1=3
"Ma2=1"
f=0.03
D=0.18

C1=sqrt(k*R*T1*1000)
Ma1=V1/C1
T01=T02
T01=T1*(1+0.5*(k-1)*Ma1^2)
T02=T2*(1+0.5*(k-1)*Ma2^2)
P01=P1*(1+0.5*(k-1)*Ma1^2)^(k/(k-1))

rho1=P1/(R*T1)
Ac=pi*D^2/4
mair=rho1*Ac*V1

P01Ps=((2+(k-1)*Ma1^2)/(k+1))^(0.5*(k+1)/(k-1))/Ma1
```

```
P1Ps=((k+1)/(2+(k-1)*Ma1^2))^0.5/Ma1
T1Ts=(k+1)/(2+(k-1)*Ma1^2)
R1Rs=((2+(k-1)*Ma1^2)/(k+1))^0.5/Ma1
V1Vs=1/R1Rs
fLs1=(1-Ma1^2)/(k*Ma1^2)+(k+1)/(2*k)*ln((k+1)*Ma1^2/(2+(k-1)*Ma1^2))
Ls1=fLs1*D/f

P02Ps=((2+(k-1)*Ma2^2)/(k+1))^(0.5*(k+1)/(k-1))/Ma2
P2Ps=((k+1)/(2+(k-1)*Ma2^2))^0.5/Ma2
T2Ts=(k+1)/(2+(k-1)*Ma2^2)
R2Rs=((2+(k-1)*Ma2^2)/(k+1))^0.5/Ma2
V2Vs=1/R2Rs
fLs2=(1-Ma2^2)/(k*Ma2^2)+(k+1)/(2*k)*ln((k+1)*Ma2^2/(2+(k-1)*Ma2^2))
Ls2=fLs2*D/f
L=Ls1-Ls2

P02=P02Ps/P01Ps*P01
P2=P2Ps/P1Ps*P1
V2=V2Vs/V1Vs*V1
```

***Discussion*** Note that the Mach number decreases nearly linearly along the duct.

---

**12-167** 

**Solution** Choked subsonic airflow in a constant cross-sectional area adiabatic duct is considered. The effect of duct length on the mass flow rate and the inlet conditions is to be investigated as the duct length is doubled.

***Assumptions*** **1** The assumptions associated with Fanno flow (i.e., steady, frictional flow of an ideal gas with constant properties through a constant cross-sectional area adiabatic duct) are valid. **2** The friction factor remains constant along the duct.

***Properties*** We take the properties of air to be $k$ = 1.4, $c_p$ = 1.005 kJ/kg·K, and $R$ = 0.287 kJ/kg·K. The average friction factor is given to be $f$ = 0.02.

***Analysis*** We use EES to solve the problem. The flow is choked, and thus $\mathrm{Ma}_2$ =1. The inlet Mach number is

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\ \mathrm{kJ/kg\cdot K})(400\ \mathrm{K})\left(\frac{1000\ \mathrm{m^2/s^2}}{1\ \mathrm{kJ/kg}}\right)} = 400.9\ \mathrm{m/s}$$

$$\mathrm{Ma}_1 = \frac{V_1}{c_1} = \frac{120\ \mathrm{m/s}}{400.9\ \mathrm{m/s}} = 0.2993$$

$\mathrm{Ma}_2 = 1$

$P_1$ = 100 kPa
$T_1$ = 400 K
$V_1$ = 120 m/s

$D$ = 6 cm

$T^*$
$P^*$
$V^*$

$L_1^*$

0 → $L$

Corresponding to this Mach number we have, from Table A-16, $fL^*/D_h$ = 5.3312, Therefore, the original duct length is

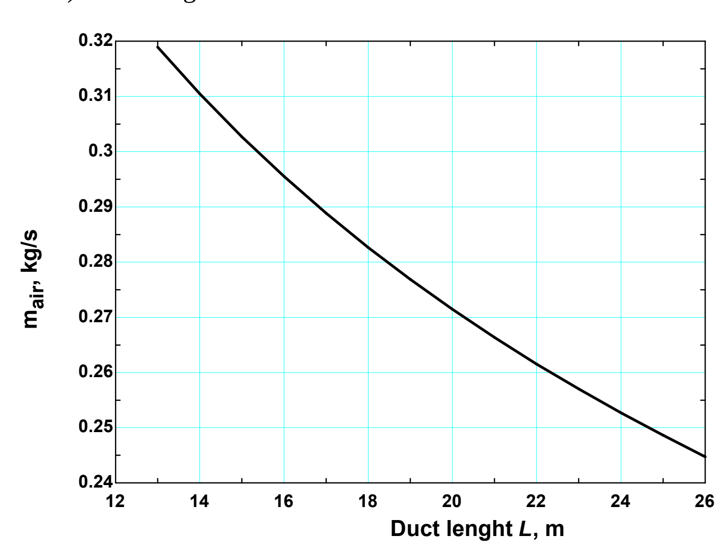
$$L = L_1^* = 5.3312\frac{D}{f} = 5.3312\frac{0.06\ \mathrm{m}}{0.02} = 16.0\ \mathrm{m}$$

Then the initial mass flow rate becomes

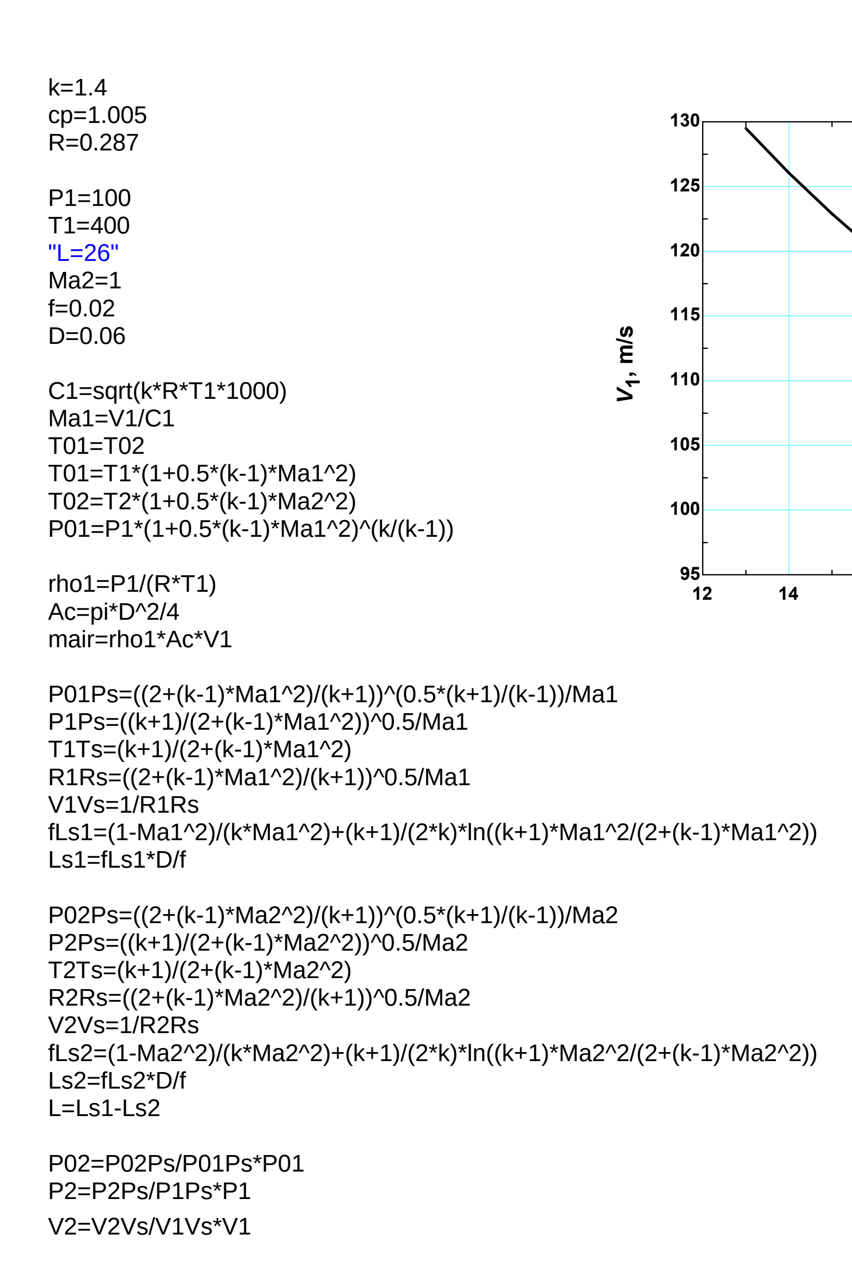
$$\rho_1 = \frac{P_1}{RT_1} = \frac{100\ \mathrm{kPa}}{(0.287\ \mathrm{kJ/kgK})(400\ \mathrm{K})} = 0.8711\ \mathrm{kg/m^3}$$

$$\dot{m}_{air} = \rho_1 A_{c1} V_1 = (0.8711\ \mathrm{kg/m^3})[\pi(0.06\ \mathrm{m})^2/4](120\ \mathrm{m/s}) = 0.296\ \mathrm{kg/s}$$

| Duct length $L$, m | Inlet velocity $V_1$, m/s | Mass flow rate $\dot{m}_{air}$, kg/s |
|---|---|---|
| 13 | 129 | 0.319 |
| 14 | 126 | 0.310 |
| 15 | 123 | 0.303 |
| 16 | 120 | 0.296 |
| 17 | 117 | 0.289 |
| 18 | 115 | 0.283 |
| 19 | 112 | 0.277 |
| 20 | 110 | 0.271 |
| 21 | 108 | 0.266 |
| 22 | 106 | 0.262 |
| 23 | 104 | 0.257 |
| 24 | 103 | 0.253 |
| 25 | 101 | 0.249 |
| 26 | 99 | 0.245 |




**The EES program is listed below, along with a plot of inlet velocity vs. duct length:**




```
k=1.4
cp=1.005
R=0.287

P1=100
T1=400
"L=26"
Ma2=1
f=0.02
D=0.06

C1=sqrt(k*R*T1*1000)
Ma1=V1/C1
T01=T02
T01=T1*(1+0.5*(k-1)*Ma1^2)
T02=T2*(1+0.5*(k-1)*Ma2^2)
P01=P1*(1+0.5*(k-1)*Ma1^2)^(k/(k-1))

rho1=P1/(R*T1)
Ac=pi*D^2/4
mair=rho1*Ac*V1

P01Ps=((2+(k-1)*Ma1^2)/(k+1))^(0.5*(k+1)/(k-1))/Ma1
P1Ps=((k+1)/(2+(k-1)*Ma1^2))^0.5/Ma1
T1Ts=(k+1)/(2+(k-1)*Ma1^2)
R1Rs=((2+(k-1)*Ma1^2)/(k+1))^0.5/Ma1
V1Vs=1/R1Rs
fLs1=(1-Ma1^2)/(k*Ma1^2)+(k+1)/(2*k)*ln((k+1)*Ma1^2/(2+(k-1)*Ma1^2))
Ls1=fLs1*D/f

P02Ps=((2+(k-1)*Ma2^2)/(k+1))^(0.5*(k+1)/(k-1))/Ma2
P2Ps=((k+1)/(2+(k-1)*Ma2^2))^0.5/Ma2
T2Ts=(k+1)/(2+(k-1)*Ma2^2)
R2Rs=((2+(k-1)*Ma2^2)/(k+1))^0.5/Ma2
V2Vs=1/R2Rs
fLs2=(1-Ma2^2)/(k*Ma2^2)+(k+1)/(2*k)*ln((k+1)*Ma2^2/(2+(k-1)*Ma2^2))
Ls2=fLs2*D/f
L=Ls1-Ls2

P02=P02Ps/P01Ps*P01
P2=P2Ps/P1Ps*P1
V2=V2Vs/V1Vs*V1
```

***Discussion*** Note that once the flow is choked, any increase in duct length results in a decrease in the mass flow rate and the inlet velocity.

**12-168**

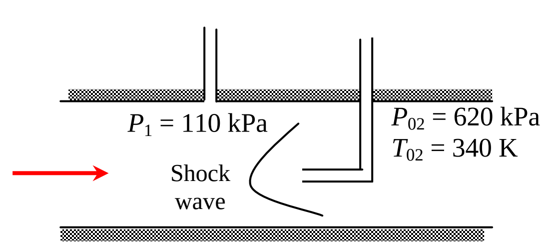
**Solution** The flow velocity of air in a channel is to be measured using a Pitot-static probe, which causes a shock wave to occur. For measured values of static pressure before the shock and stagnation pressure and temperature after the shock, the flow velocity before the shock is to be determined.

***Assumptions*** **1** Air is an ideal gas with constant specific heats. **2** Flow through the nozzle is steady and one-dimensional.

***Properties*** The specific heat ratio of air at room temperature is $k$= 1.4.

***Analysis*** The nose of the probe is rounded (instead of being pointed), and thus it will cause a bow shock wave to form. Bow shocks are difficult to analyze. But they are normal to the body at the nose, and thus we can approximate them as normal shocks in the vicinity of the probe. It is given that the static pressure before the shock is $P_1$ = 110 kPa, and the stagnation pressure and temperature after the shock are $P_{02}$ = 620 kPa, and $T_{02}$ = 340 K. Noting that the stagnation temperature remains constant, we have

$$T_{01} = T_{02} = 340\text{ K}$$




Also, $\dfrac{P_{02}}{P_1} = \dfrac{620\text{ kPa}}{110\text{ kPa}} = 5.6364 \approx 5.64$

The fluid properties after the shock are related to those before the shock through the functions listed in Table A-14. For $P_{02} / P_1 = 5.64$ we read

$$\text{Ma}_1 = 2.0,\quad \text{Ma}_2 = 0.5774,\quad \frac{P_{02}}{P_{01}} = 0.7209,\quad \frac{V_1}{V_2} = \frac{\rho_2}{\rho_1} = 2.6667,$$

Then the stagnation pressure and temperature before the shock become

$P_{01}$ = $P_{02}$ /0.7209 = (620 kPa)/0.7209 = 860 kPa

$$T_1 = T_{01}\left(\frac{P_1}{P_{01}}\right)^{(k-1)/k} = (340\text{ K})\left(\frac{110\text{ kPa}}{860\text{ kPa}}\right)^{(1.4-1)/1.4} = 188.9\text{ K}$$

The flow velocity before the shock can be determined from $V_1$ = Ma$_1c_1$, where $c_1$ is the speed of sound before the shock,

$$c_1 = \sqrt{kRT_1} = \sqrt{(1.4)(0.287\text{ kJ/kg}\cdot\text{K})(188.9\text{ K})\left(\frac{1000\text{ m}^2/\text{s}^2}{1\text{ kJ/kg}}\right)} = 275.5\text{ m/s}$$

$$V_1 = \text{Ma}_1 c_1 = 2(275.5\text{ m/s}) = \mathbf{551\ m/s}$$

***Discussion*** The flow velocity after the shock is $V_2$ = $V_1$/2.6667 = 551/2.6667 = 207 m/s. Therefore, the velocity measured by a Pitot-static probe would be very different that the flow velocity.

---

## Design and Essay Problems

**12-169 to 12-171**

**Solution** Students' essays and designs should be unique and will differ from each other.



# Solutions Manual for
# Fluid Mechanics: Fundamentals and Applications
# by Çengel & Cimbala

---

# CHAPTER 13
# OPEN-CHANNEL FLOW

---

## Classification, Froude Number, and Wave Speed

### 13-1C

**Solution** We are to discuss differences between open-channel flow and internal flow.

***Analysis*** *Open-channel flow* is the **flow of liquids in channels open to the atmosphere or in partially filled conduits**, and is characterized by the presence of a **liquid-gas interface called the *free surface***, whereas *internal flow* is the **flow of liquids or gases that completely fill a conduit**.

***Discussion*** At a free surface of a liquid, the pressure must be equal to the pressure of the gas above it. This is what controls the pressure in open-channel flow. There is no such restriction in internal (pipe) flow since there is no free surface.

### 13-2C

**Solution** We are to discuss the driving force in open-channel flow and how flow rate is determined.

***Analysis*** Flow in a channel is driven **naturally by gravity**. Water flow in a river, for example, is driven by the elevation difference between the source and the sink. The flow rate in an open channel is established by the **dynamic balance between gravity and friction**. Inertia of the flowing fluid also becomes important in unsteady flow.

***Discussion*** In pipe flow, on the other hand, there may be an additional driving force of pressure due to pumps.

### 13-3C

**Solution** We are to discuss how pressure changes along the free surface in open-channel flow.

***Analysis*** The free surface **coincides with the hydraulic grade line** (HGL), and the **pressure is constant along the free surface**.

***Discussion*** At a free surface of a liquid, the pressure must be equal to the pressure of the gas above it.

### 13-4C

**Solution** We are to determine if the slope of the free surface is equal to the slope of the channel bottom.

***Analysis*** **No in general**. The slope of the free surface is **not necessarily equal** to the slope of the bottom surface even during steady fully developed flow.

***Discussion*** However, there are situations called *uniform flow* in which the conditions here are met.

### 13-5C

**Solution** We are to discuss the difference between uniform and nonuniform flow.

***Analysis*** The flow in a channel is said to be *uniform* if the **flow depth (and thus the average velocity) remains constant**. Otherwise, the flow is said to be *nonuniform* or *varied*, indicating that the flow depth varies with distance in the flow direction. Uniform flow conditions are commonly encountered in practice in long straight sections of channels with constant slope and constant cross-section.

***Discussion*** In uniform open-channel flow, the head loss due to frictional effects equals the elevation drop.

**13-6C**

**Solution** We are to define normal depth and how it is established.

***Analysis*** In open channels of constant slope and constant cross-section, the fluid accelerates until the head loss due to frictional effects equals the elevation drop. The fluid at this point reaches its terminal velocity, and uniform flow is established. The flow remains uniform as long as the slope, cross-section, and the surface roughness of the channel remain unchanged. **The flow depth in uniform flow** is called the *normal depth* $y_n$, which is an important characteristic parameter for open-channel flows.

***Discussion*** The normal depth is a fairly strong function of surface roughness.

---

**13-7C**

**Solution** We are to discuss some reasons for nonuniform flow in open channels, and the difference between rapidly varied flow and gradually varied flow.

***Analysis*** The **presence of an obstruction in a channel such as a gate or a change in slope or cross-section** causes the flow depth to vary, and thus the flow to become varied or nonuniform. The varied flow is called *rapidly varied flow* (RVF) if the flow depth changes markedly over a relatively short distance in the flow direction (such as the flow of water past a partially open gate or shortly before a falls), and *gradually varied flow* (GVF) if the flow depth changes gradually over a long distance along the channel.

***Discussion*** The equations of GVF are simplified because of the slow changes in the flow direction.

---

**13-8C**

**Solution** We are to define and discuss hydraulic radius.

***Analysis*** The *hydraulic radius* $R_h$ is defined as **the ratio of the cross-sectional flow area $A_c$ and the wetted perimeter $p$**. That is, $R_h = A_c/p$. Knowing the hydraulic radius, the hydraulic diameter is determined from $D_h = 4R_h$.

***Discussion*** It is unfortunate (and not our fault!) that hydraulic radius is ¼ rather than ½ of hydraulic diameter.

---

**13-9C**

**Solution** We are to explain how to determine if a flow is tranquil, critical, or rapid.

***Analysis*** Knowing the average flow velocity and flow depth, the Froude number is determined from $\mathrm{Fr} = V/\sqrt{gy}$. Then the flow is classified as

**Fr < 1** **Subcritical or tranquil flow**
**Fr = 1** **Critical flow**
**Fr > 1** **Supercritical or rapid flow**

***Discussion*** The Froude number is the most important parameter in open-channel flow.

---

**13-10C**

**Solution** We are to define and discuss the usefulness of the Froude number.

***Analysis*** *Froude number*, defined as $\boxed{\mathrm{Fr} = V/\sqrt{gy}}$, is a **dimensionless parameter that governs the character of flow in open channels**. Here, $g$ is the gravitational acceleration, $V$ is the mean fluid velocity at a cross-section, and $L_c$ is a characteristic length ($L_c$ = flow depth $y$ for wide rectangular channels). Fr represents the **ratio of inertia forces to viscous forces in open-channel flow**. The Froude number is also the **ratio of the flow speed to wave speed**, $\mathrm{Fr} = V/c_o$.

***Discussion*** The Froude number is the most important parameter in open-channel flow.

---

**13-11C**

**Solution** We are to define critical length, and discuss how it is determined.

***Analysis*** The **flow depth $y_c$ corresponding to a Froude number of Fr = 1** is the *critical depth*, and it is determined from $\boxed{V = \sqrt{g y_c}}$ or $\boxed{y_c = V^2 / g}$.

***Discussion*** Critical depth is a useful parameter, even if the depth does not actually equal $y_c$ anywhere in the flow.

---

**13-12C**

**Solution** We are to discuss whether the flow upstream of a hydraulic jump must be supercritical, and whether the flow downstream of a hydraulic jump must be subcritical.

***Analysis*** Upstream of a hydraulic jump, the **upstream flow must be supercritical**. Downstream of a hydraulic jump, the **downstream flow must be subcritical**.

***Discussion*** Otherwise, the second law of thermodynamics would be violated. Note that a hydraulic jump is analogous to a normal shock wave – in that case, the flow upstream must be supersonic and the flow downstream must be subsonic.

---

**13-13**

**Solution** The flow of water in a wide channel is considered. The speed of a small disturbance in flow for two different flow depths is to be determined for both water and oil.

***Assumptions*** The distance across the wave is short and thus friction at the bottom surface and air drag at the top are negligible,

***Analysis*** Surface wave speed can be determined directly from the relation $c_0 = \sqrt{gh}$ .

(a) $c_0 = \sqrt{gh} = \sqrt{(9.81\,\text{m/s}^2)(0.1\,\text{m})} = \mathbf{0.990\ m/s}$

(b) $c_0 = \sqrt{gh} = \sqrt{(9.81\,\text{m/s}^2)(0.8\,\text{m})} = \mathbf{2.80\ m/s}$

Therefore, a disturbance in the flow will travel at a speed of 0.990 m/s in the first case, and 2.80 m/s in the second case.

***Discussion*** Note that wave speed depends on the water depth, and the wave speed increases as the water depth increases as long as the water remains shallow. Results would not change if the fluid were oil, because the wave speed depends only on the fluid depth.

---

**13-14**

**Solution** Water flows uniformly in a wide rectangular channel. For given flow depth and velocity, it is to be determined whether the flow is laminar or turbulent, and whether it is subcritical or supercritical.

***Assumptions*** The flow is uniform.

***Properties*** The density and dynamic viscosity of water at 20ºC are $\rho = 998.0$ kg/m$^3$ and $\mu = 1.002\times10^{-3}$ kg/m·s.

***Analysis*** (a) The Reynolds number of the flow is $\text{Re} = \dfrac{\rho V y}{\mu} = \dfrac{(998.0\,\text{kg/m}^3)(2\,\text{m/s})(0.2\,\text{m})}{1.002\times10^{-3}\,\text{kg/m}\cdot\text{s}} = 3.984\times10^5$ , which is greater than the critical value of 500. Therefore, the flow is **turbulent**.

(*b*) The Froude number is $\text{Fr} = \dfrac{V}{\sqrt{gy}} = \dfrac{2\,\text{m/s}}{\sqrt{(9.81\,\text{m/s}^2)(0.2\,\text{m})}} = 1.43$, which is greater than 1.

Therefore, the flow is **supercritical**.

***Discussion*** The result in (*a*) is expected since almost all open channel flows are turbulent. Also, hydraulic radius for a wide rectangular channel approaches the water depth *y* as the ratio *y*/b approaches zero.

---

## 13-15

**Solution** Water flow in a partially full circular channel is considered. For given water depth and average velocity, the hydraulic radius, Reynolds number, and the flow regime are to be determined.

***Assumptions*** **1** The flow is uniform.

***Properties*** The density and dynamic viscosity of water at 20ºC are $\rho = 998.0$ kg/m$^3$ and $\mu = 1.002\times10^{-3}$ kg/m·s.

***Analysis*** From geometric considerations,

$$\cos\theta = \frac{R-a}{R} = \frac{1-0.5}{1} = 0.5 \quad \rightarrow \quad \theta = 60° = 60\frac{2\pi}{360} = \frac{\pi}{3}$$

Then the hydraulic radius becomes

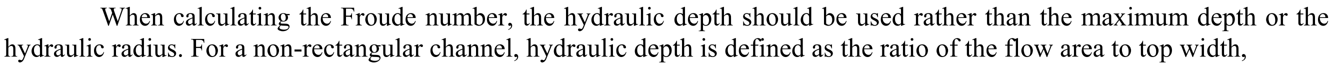
$$R_h = \frac{A_c}{p} = \frac{\theta - \sin\theta\cos\theta}{2\theta}R = \frac{\pi/3 - \sin(\pi/3)\cos(\pi/3)}{2\pi/3}(1\text{ m}) = \mathbf{0.293\ m}$$

The Reynolds number of the flow is

$$\text{Re} = \frac{\rho V R_h}{\mu} = \frac{(998.0\text{ kg/m}^3)(2\text{ m/s})(0.293\text{ m})}{1.002\times10^{-3}\text{ kg/m}\cdot\text{s}} = \mathbf{5.84\times10^5}$$

which is greater than the critical value of 500. Therefore, the flow is turbulent.

When calculating the Froude number, the hydraulic depth should be used rather than the maximum depth or the hydraulic radius. For a non-rectangular channel, hydraulic depth is defined as the ratio of the flow area to top width,



$$A_c = R^2(\theta - \sin\theta\cos\theta) = (1\text{ m})^2[\pi/3 - \sin(\pi/3)\cos(\pi/3)] = 0.6142\text{ m}^2$$

$$y_h = \frac{A_c}{\text{Top width}} = \frac{A_c}{2R\sin\theta} = \frac{0.6142\text{ m}^2}{2(1\text{ m})\sin 60°} = 0.3546\text{ m} \quad \rightarrow \quad \text{Fr} = \frac{V}{\sqrt{g y_h}} = \frac{2\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(0.3546\text{ m})}} = 1.07$$

which is greater than 1. Therefore, the flow is **supercritical** (although, very close to critical).

***Discussion*** Note that if the maximum flow depth were used instead of the hydraulic depth, the result would be subcritical flow, which is not true.

---

## 13-16

**Solution** Water flows uniformly in a wide rectangular channel. For given values of flow depth and velocity, it is to be determined whether the flow is subcritical or supercritical.

***Assumptions*** **1** The flow is uniform. **2** The channel is wide and thus the side wall effects are negligible.

***Analysis*** The Froude number is $\text{Fr} = \frac{V}{\sqrt{gy}} = \frac{4\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(0.08\text{ m})}} = 4.51$, which is greater than 1.

Therefore, the flow is **supercritical**.

***Discussion*** Note that the Froude Number is not function of any temperature-dependent properties, and thus temperature.

---

## 13-17

**Solution** Rain water flows on a concrete surface. For given values of flow depth and velocity, it is to be determined whether the flow is subcritical or supercritical.

***Assumptions*** **1** The flow is uniform. **2** The thickness of water layer is constant.

***Analysis*** The Froude number is $\text{Fr} = \frac{V}{\sqrt{gy}} = \frac{1.3\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(0.02\text{ m})}} = 2.93$, which is greater than 1.

Therefore, the flow is **supercritical**.

***Discussion*** This water layer will undergo a hydraulic jump when the ground slope decreases or becomes adverse.

**13-18E**

**Solution** Water flows uniformly in a wide rectangular channel. For given flow depth and velocity, it is to be determined whether the flow is laminar or turbulent, and whether it is subcritical or supercritical.

***Assumptions*** The flow is uniform.

***Properties*** The density and dynamic viscosity of water at 70ºF are $\rho$ = 62.30 lbm/ft$^3$ and $\mu$ = 6.556×10$^{-4}$ lbm/ft·s.

***Analysis*** (*a*) The Reynolds number of the flow is $\text{Re} = \frac{\rho V y}{\mu} = \frac{(62.30\ \text{lbm/ft}^3)(6\ \text{ft/s})(0.5\ \text{ft})}{6.556\times10^{-4}\ \text{lbm/ft}\cdot\text{s}} = 2.85\times10^5$, which is greater than the critical value of 500. Therefore, the flow is **turbulent**.

(*b*) The Froude number is $\text{Fr} = \frac{V}{\sqrt{gy}} = \frac{6\ \text{ft/s}}{\sqrt{(32.2\ \text{ft/s}^2)(0.5\ \text{ft})}} = 1.50$, which is greater than 1.

Therefore, the flow is **supercritical**.

***Discussion*** The result in (*a*) is expected since almost all open channel flows are turbulent. Also, hydraulic radius for a wide rectangular channel approaches the water depth *y* as the ratio *y*/b approaches zero.

---

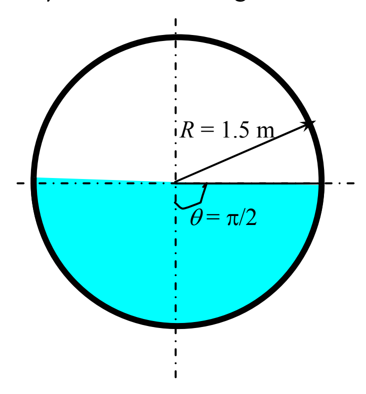
**13-19**

**Solution** Water flows uniformly through a half-full circular channel. For a given average velocity, the hydraulic radius, the Reynolds number, and the flow regime are to be determined.

***Assumptions*** The flow is uniform.

***Properties*** The density and dynamic viscosity of water at 10ºC are $\rho$ = 999.7 kg/m$^3$ and $\mu$ = 1.307×10$^{-3}$ kg/m·s.

***Analysis*** From geometric considerations, the hydraulic radius is




$$R_h = \frac{A_c}{p} = \frac{\pi R^2/2}{\pi R} = \frac{R}{2} = \frac{1.5\ \text{m}}{2} = \mathbf{0.75\ m}$$

The Reynolds number of the flow is

$$\text{Re} = \frac{\rho V R_h}{\mu} = \frac{(999.7\ \text{kg/m}^3)(2.5\ \text{m/s})(0.75\ \text{m})}{1.307\times10^{-3}\ \text{kg/m}\cdot\text{s}} = \mathbf{1.43\times10^6}$$, which is

greater than the critical value of 500. Therefore, **the flow is turbulent**.

When calculating the Froude number, the hydraulic depth should be used rather than the maximum depth or the hydraulic radius. For a non-rectangular channel, hydraulic depth is defined as the ratio of the flow area to top width,

$$y_h = \frac{A_c}{\text{Top width}} = \frac{\pi R^2/2}{2R} = \frac{\pi R}{4} = \frac{\pi(1.5\ \text{m})}{4} = 1.178\ \text{m}$$

$$\text{Fr} = \frac{V}{\sqrt{g y_h}} = \frac{2.5\ \text{m/s}}{\sqrt{(9.81\ \text{m/s}^2)(1.178\ \text{m})}} = 0.735$$, which is greater than 1. Therefore, the flow is **subcritical**.

***Discussion*** If the maximum flow depth were used instead of the hydraulic depth, the result would still be subcritical flow, but this is not always the case.

---

**13-20**

**Solution** A single wave is initiated in a sea by a strong jolt during an earthquake. The speed of the resulting wave is to be determined.

***Assumptions*** The depth of water is constant,

***Analysis*** Surface wave speed is determined the wave-speed relation to be

$$c_0 = \sqrt{gh} = \sqrt{(9.81\,\text{m/s}^2)(2000\,\text{m})} = \mathbf{140\,m/s}$$

***Discussion*** Note that wave speed depends on the water depth, and the wave speed increases as the water depth increases. Also, the waves eventually die out because of the viscous effects.

---

## Specific Energy and the Energy Equation

**13-21C**

**Solution** We are to define and discuss specific energy.

***Analysis*** The *specific energy* $E_s$ of a fluid flowing in an open channel is the **sum of the pressure and dynamic heads of a fluid**, and is expressed as $\boxed{E_s = y + \frac{V^2}{2g}}$.

***Discussion*** Specific energy is very useful when analyzing varied flows.

---

**13-22C**

**Solution** We are to compare the specific energy in two flows – one subcritical and one supercritical.

***Analysis*** A plot of $E_s$ versus $y$ for constant $\dot{V}$ through a rectangular channel of width $b$ reveals that there are two $y$ values corresponding to a fixed value of $E_s$: one for subcritical flow and one for supercritical flow. Therefore, **the specific energies of water in those two channels can be identical**.

***Discussion*** If the flow is varied (not uniform), however, $E_s$ is not *necessarily* identical in the two channels.

---

**13-23C**

**Solution** We are to examine claims about the minimum value of specific energy.

***Analysis*** The point of minimum specific energy is the critical point, and thus **the first person is correct**.

***Discussion*** The specific energy cannot go below the critical point for a given volume flow rate, as is clear from the plot of specific energy as a function of flow depth.

---

**13-24C**

**Solution** We are to examine a claim about supercritical flow of water in an open channel, namely, that the larger the flow depth, the larger the specific energy.

***Analysis*** **No, the claim is incorrect**. A plot of $E_s$ versus $y$ for constant $\dot{V}$ reveals that the *specific energy decreases as the flow depth increases during supercritical channel flow*.

***Discussion*** This may go against our intuition, since a larger flow depth seems to imply greater energy, but this is not necessarily the case (we cannot always trust our intuition).

---

**13-25C**

**Solution** We are to examine a claim that specific energy remains constant in steady uniform flow.

***Analysis*** The **first person (who claims that specific energy remains constant) is correct** since in uniform flow, the flow depth and the flow velocity, and thus the specific energy, remain constant since $E_s = y + V^2/2g$. The head loss is made up by the decline in elevation (the channel is sloped downward in the flow direction).

***Discussion*** In uniform flow, the flow depth and the average velocity do not change downstream, since the elevation drop exactly overcomes the frictional losses.

---

**13-26C**

**Solution** We are to define and discuss friction slope.

***Analysis*** The *friction slope* is related to head loss $h_L$, and is defined as $\boxed{S_f = h_L / L}$ where $L$ is the channel length. The **friction slope is equal to the bottom slope when the head loss is equal to the elevation drop**. That is, $S_f = S_0$ when $h_L = z_1 - z_2$.

***Discussion*** Friction slope is a useful concept when analyzing uniform or varied flow in open channels.

---

**13-27C**

**Solution** We are to examine a claim that during steady flow in a wide rectangular channel, the energy line of the flow is parallel to the channel bottom when the frictional losses are negligible.

***Analysis*** **No, the claim is not correct**. The energy line is a distance $E_s = y + V^2/2g$ (total mechanical energy of the fluid) above a horizontal reference datum. When there is no head loss, the energy line is horizontal even when the channel is not. The elevation and velocity heads ($z + y$ and $V^2/2g$) may convert to each other during flow in this case, but their sum remains constant.

***Discussion*** Keep in mind that in real life, there is no such thing as frictionless flow. However, there are situations in which the frictional effects are negligible compared to other effects in the flow.

---

**13-28C**

**Solution** We are to examine a claim that during steady 1-D flow through a wide rectangular channel, the total mechanical energy of the fluid at the free surface is equal to that of the fluid at the channel bottom.

***Analysis*** **Yes, the claim is correct**. During steady one-dimensional flow, the total mechanical energy of a fluid at any point of a cross-section is given by $H = z + y + V^2/2g$.

***Discussion*** The physical elevation of the point under consideration does not appear in the above equation for *H*.

---

**13-29C**

**Solution** We are to express the total mechanical energy in steady 1-D flow in terms of heads.

***Analysis*** The total mechanical energy of a fluid at any point of a cross-section is expressed as $\boxed{H = z + y + V^2/2g}$ where *y* is the flow depth, *z* is the elevation of the channel bottom, and *V* is the average flow velocity. It is related to the specific energy of the fluid by $H = z + E_s$.

***Discussion*** Because of irreversible frictional head losses, *H* must decrease in the flow direction in open-channel flow.

---

**13-30C**

**Solution** We are to express the 1-D energy equation for open-channel flow and discuss head loss.

***Analysis*** The one-dimensional energy equation for open channel flow between an upstream section 1 and downstream section 2 is written as $\boxed{z_1 + y_1 + \frac{V_1^2}{2g} = z_2 + y_2 + \frac{V_2^2}{2g} + h_L}$ where $y$ is the flow depth, $z$ is the elevation of the channel bottom, and $V$ is the average flow velocity. The head loss $h_L$ due to frictional effects can be determined from $\boxed{h_L = f\frac{L}{R_h}\frac{V^2}{8g}}$ where $f$ is the average friction factor and $L$ is the length of channel between sections 1 and 2.

***Discussion*** Head loss is always positive – it can never be negative since this would violate the second law of thermodynamics. Thus, the total mechanical energy must decrease downstream in open-channel flow.

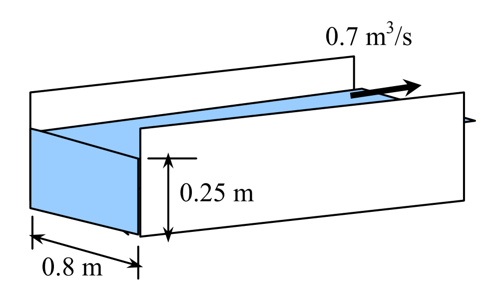
---

**13-31**

**Solution** Water flow in a rectangular channel is considered. The character of flow, the flow velocity, and the alternate depth are to be determined.

***Assumptions*** The specific energy is constant.

***Analysis*** The average flow velocity is determined from




$$V = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{yb} = \frac{0.7\text{ m}^3/\text{s}}{(0.25\text{ m})(0.8\text{ m})} = \mathbf{3.50\text{ m/s}}$$

The critical depth for this flow is

$$y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{(0.7\text{ m}^3/\text{s})^2}{(9.81\text{ m/s}^2)(0.8\text{ m})^2}\right)^{1/3} = 0.427\text{ m}$$

Therefore, the flow is *supercritical* since the actual flow depth is $y = 0.25$ m, and $y < y_c$. The specific energy for given conditions is

$$E_{s1} = y_1 + \frac{\dot{V}^2}{2gb^2y_1^2} = (0.25\text{ m}) + \frac{(0.7\text{ m}^3/\text{s})^2}{2(9.81\text{ m/s}^2)(0.8\text{ m})^2(0.25\text{ m})^2} = 0.874\text{ m}$$

Then the alternate depth is determined from $E_{s1} = E_{s2}$ to be

$$E_{s2} = y_2 + \frac{\dot{V}^2}{2gb^2y_2^2} \quad \rightarrow \quad 0.874\text{ m} = y_2 + \frac{(0.7\text{ m}^3/\text{s})^2}{2(9.81\text{ m/s}^2)(0.8\text{ m})^2 y_2^2}$$

Solving for $y_2$ gives the alternate depth to be $y_2$ = **0.82 m**. There are three roots of this equation; one for subcritical, one for supercritical and third one as a negative root. Therefore, if the character of flow is changed from supercritical to subcritical while holding the specific energy constant, the flow depth will rise from 0.25 m to 0.82 m.

***Discussion*** Two alternate depths show two possible flow conditions for a given specific energy. If the energy is not the minimum specific energy, there are two water depths corresponding to subcritical and supercritical states of flow. As an example, these two depths may be observed before and after a sluice gate as alternate depths, if the losses are disregarded.

---

**13-32**

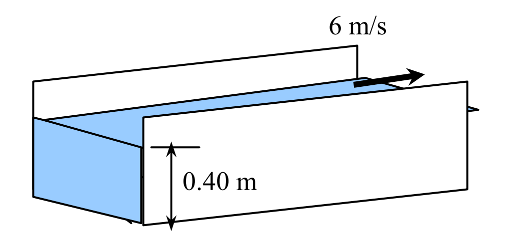
**Solution** Water flows in a rectangular channel. The specific energy and whether the flow is subcritical or supercritical are to be determined.

***Assumptions*** The flow is uniform and thus the specific energy is constant.

***Analysis*** For convenience, we take the channel width to be $b$ = 1 m. Then the volume flow rate and the critical depth for this flow become

$$\dot{V} = VA_c = Vyb = (6\text{ m/s})(0.4\text{ m})(1\text{ m}) = 2.40\text{ m}^3\text{/s}$$

$$y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{(2.40\text{ m}^3\text{/s})^2}{(9.81\text{ m/s}^2)(1\text{ m})^2}\right)^{1/3} = \mathbf{0.837\text{ m}}$$




The flow is **supercritical** since the actual flow depth is $y$ = 0.4 m, and $y < y_c$. The specific energy for given conditions is

$$E_{s1} = y_1 + \frac{\dot{V}^2}{2gb^2y_1^2} = y_1 + \frac{V^2}{2g} = (0.4\text{ m}) + \frac{(6\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = \mathbf{2.23\text{ m}}$$

***Discussion*** Note that the flow may also exist as subcritical flow at the same value of specific energy,

**13-33** [*Also solved using EES on enclosed DVD*]

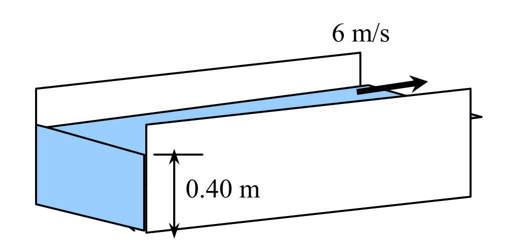
**Solution** Water flows in a rectangular channel. The critical depth, the alternate depth, and the minimum specific energy are to be determined.

***Assumptions*** The channel is sufficiently wide so that the edge effects are negligible.

***Analysis*** For convenience, we take the channel width to be $b$ = 1 m. Then the volume flow rate and the critical depth for this flow become

$$\dot{V} = VA_c = Vyb = (6\text{ m/s})(0.4\text{ m})(1\text{ m}) = 2.40\text{ m}^3\text{/s}$$

$$y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{(2.40\text{ m}^3\text{/s})^2}{(9.81\text{ m/s}^2)(1\text{ m})^2}\right)^{1/3} = \mathbf{0.837\text{ m}}$$




(*b*) The flow is *supercritical* since the actual flow depth is $y$ = 0.4 m, and $y < y_c$. The specific energy for given conditions is

$$E_{s1} = y_1 + \frac{\dot{V}^2}{2gb^2y_1^2} = y_1 + \frac{V^2}{2g} = (0.4\text{ m}) + \frac{(6\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 2.23\text{ m}$$

Then the alternate depth is determined from $E_{s1} = E_{s2}$ to be

$$E_{s2} = y_2 + \frac{\dot{V}^2}{2gb^2y_2^2} \quad \to \quad 2.23\text{ m} = y_2 + \frac{0.240\text{ m}^3\text{/s}}{y_2^2}$$

Solving for $y_2$ gives the alternate depth to be $y_2$ = **2.17 m**. Therefore, if the character of flow is changed from supercritical to subcritical while holding the specific energy constant, the flow depth will rise from 0.4 m to 2.17 m.

(*c*) the minimum specific energy is

$$E_{s,\min} = y_c + \frac{V_c^2}{2g} = y_c + \frac{gy_c}{2g} = \frac{3}{2}y_c = \frac{3}{2}(0.837\text{ m}) = \mathbf{1.26\text{ m}}$$

***Discussion*** Note that minimum specific energy is observed when the flow depth is critical.

## 13-34

**Solution** Water flows in a rectangular channel. The critical depth, the alternate depth, and whether the flow is subcritical or supercritical are to be determined.

***Assumptions*** The flow is uniform and thus the specific energy is constant.

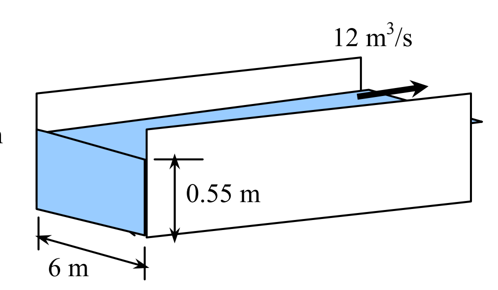
***Analysis*** (*a*) The critical depth is calculated to be $y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{(12\text{ m}^3/\text{s})^2}{(9.81\text{ m/s}^2)(6\text{ m})^2}\right)^{1/3} = 0.742\text{ m}$

(*b*) The average flow velocity and the Froude number are

$$V = \frac{\dot{V}}{b\,y} = \frac{12\text{ m}^3/\text{s}}{(6\text{ m})(0.55\text{ m})} = 3.636\text{ m/s} \quad \text{and} \quad \text{Fr}_1 = \frac{V}{\sqrt{gy}} = \frac{3.636\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(0.55\text{ m})}} = 1.565$$

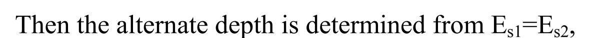
, which is greater than 1.

Therefore, the flow is **supercritical**.




(*c*) Specific energy for this flow is

$$E_{s1} = y_1 + \frac{\dot{V}^2}{2gb^2y_1^2} = (0.55\text{ m}) + \frac{(12\text{ m}^3/\text{s})^2}{2(9.81\text{ m/s}^2)(6\text{ m})^2(0.55\text{ m})^2} = 1.224\text{ m}$$



Then the alternate depth is determined from $E_{s1}=E_{s2}$,

$$E_{s2} = y_2 + \frac{\dot{V}^2}{2gb^2y_2^2} \quad \rightarrow \quad 1.224\text{ m} = y_2 + \frac{(12\text{ m}^3/\text{s})^2}{2(9.81\text{ m/s}^2)(6\text{ m})^2 y_2^2}$$

The alternate depth is calculated to be $y_2 =$ **1.03 m** which is the subcritical depth for the same value of specific energy.

***Discussion*** The depths 0.55 m and 1.03 are alternate depths for the given discharge and specific energy. The flow conditions determine which one is observed.

---

## 13-35E

**Solution** Water flows in a wide rectangular channel. For specified values of flow depth and average velocity, the Froude number, critical depth, and whether the flow is subcritical or supercritical are to be determined.

***Assumptions*** The flow is uniform and thus the specific energy is constant.

***Analysis*** (*a*) The Froude number is $\text{Fr} = \frac{V}{\sqrt{gy}} = \frac{14\text{ ft/s}}{\sqrt{(32.2\text{ ft/s}^2)(0.8\text{ ft})}} = \mathbf{2.76}$

(*b*) The critical depth is calculated to be $y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{V^2y^2b^2}{gb^2}\right)^{1/3} = \left(\frac{(14\text{ ft/s})^2(0.8\text{ ft})^2}{(32.2\text{ ft/s}^2)}\right)^{1/3} = \mathbf{1.57\ ft}$

(*c*) The flow is **supercritical** since Fr > 1.

**For the case of *y* = 0.2 ft:**
Replacing 0.8 ft in above calculations by 0.2 ft gives

14 ft/s

0.8 ft

$$\text{Fr} = \frac{V}{\sqrt{gy}} = \frac{14\text{ ft/s}}{\sqrt{(32.2\text{ ft/s}^2)(0.2\text{ ft})}} = \mathbf{5.52}$$

$$y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{V^2y^2b^2}{gb^2}\right)^{1/3} = \left(\frac{(14\text{ ft/s})^2(0.2\text{ ft})^2}{(32.2\text{ ft/s}^2)}\right)^{1/3} = \mathbf{0.625\ ft}$$

The flow is supercritical in this case also since Fr > 1.

***Discussion*** Note that the value of critical depth depends on flow rate, and it decreases as the flow rate decreases.

---

**13-36E**

**Solution** Water flows in a wide rectangular channel. For specified values of flow depth and average velocity, the Froude number, critical depth, and whether the flow is subcritical or supercritical are to be determined.

***Assumptions*** The flow is uniform and thus the specific energy is constant.

***Analysis*** (*a*) The Froude number is $\mathrm{Fr} = \dfrac{V}{\sqrt{gy}} = \dfrac{10\ \mathrm{ft/s}}{\sqrt{(32.2\ \mathrm{ft/s^2})(0.8\ \mathrm{ft})}} = \mathbf{1.97}$

(*b*) The critical depth is calculated to be $y_c = \left(\dfrac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\dfrac{V^2 y^2 b^2}{gb^2}\right)^{1/3} = \left(\dfrac{(10\ \mathrm{ft/s})^2 (0.8\ \mathrm{ft})^2}{(32.2\ \mathrm{ft/s^2})}\right)^{1/3} = \mathbf{1.26\ ft}$

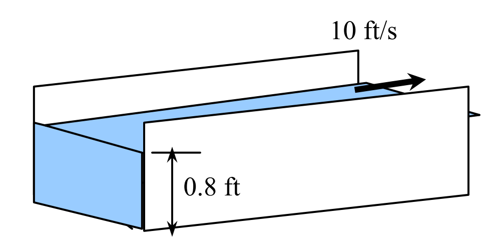
(*c*) The flow is **supercritical** since Fr > 1.

**For the case of $y$ = 0.2 ft:**
Replacing 0.8 ft in above calculations by 0.2 ft gives

$$\mathrm{Fr} = \frac{V}{\sqrt{gy}} = \frac{10\ \mathrm{ft/s}}{\sqrt{(32.2\ \mathrm{ft/s^2})(0.2\ \mathrm{ft})}} = \mathbf{3.94}$$

$$y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{V^2 y^2 b^2}{gb^2}\right)^{1/3} = \left(\frac{(14\ \mathrm{ft/s})^2 (0.2\ \mathrm{ft})^2}{(32.2\ \mathrm{ft/s^2})}\right)^{1/3} = \mathbf{0.50\ ft}$$

The flow is supercritical in this case also since Fr > 1.




***Discussion*** Note that the value of critical depth depends on flow rate, and it decreases as the flow rate decreases.

---

**13-37**

**Solution** Critical flow of water in a rectangular channel is considered. For a specified average velocity, the flow rate of water is to be determined.

***Assumptions*** The flow is uniform and thus the specific energy is constant.

***Analysis*** The Froude number must be unity since the flow is critical, and thus $\mathrm{Fr} = V/\sqrt{gy} = 1$. Therefore,

$$y = y_c = \frac{V^2}{g} = \frac{(5\ \mathrm{m/s})^2}{9.81\ \mathrm{m/s^2}} = \mathbf{2.55\ m}$$

Then the flow rate becomes

$$\dot{V} = VA_c = Vby = (5\ \mathrm{m/s})(4\ \mathrm{m})(2.55\ \mathrm{m}) = \mathbf{51.0\ m^3/s}$$

***Discussion*** Critical flow is not a stable type of flow and can be observed for short intervals. Occurrence of critical depth is important as boundary condition most of the time. For example it can be used as a flow rate computation mechanism for a channel ending with a drawdown.

---

**13-38**

**Solution** Water flows uniformly through a half-full circular steel channel. For a given average velocity, the volume flow rate, critical slope, and the critical depth are to be determined.

***Assumptions*** The flow is uniform.

***Analysis*** The volume flow rate is determined from

$$\dot{V} = VA_c = V\frac{\pi R^2}{2} = (2.8\ \text{m/s})\frac{\pi(0.25\ \text{m})^2}{2} = \mathbf{0.275\ m^3/s}$$

When calculating the Froude number, the hydraulic depth should be used rather than the maximum depth or the hydraulic radius. For a non-rectangular channel, hydraulic depth is defined as the ratio of the flow area to top width,

$$y_h = \frac{A_c}{\text{Top width}} = \frac{\pi R^2/2}{2R} = \frac{\pi R}{4} = \frac{\pi(0.25\ \text{m})}{4} = 0.1963\ \text{m}$$

$$\text{Fr} = \frac{V}{\sqrt{gy}} = \frac{2.8\ \text{m/s}}{\sqrt{(9.81\ \text{m/s}^2)(0.1963\ \text{m})}} = 2.02$$

which is greater than 1. Therefore, the flow is **supercritical**.

***Discussion*** Note that if the maximum flow depth were used instead of the hydraulic depth, the result could be different, especially when the Froude number is close to 1.

---

**13-39**

**Solution** Water flows uniformly through a half-full hexagon channel. For a given flow rate, the average velocity and whether the flow is subcritical or supercritical are to be determined.

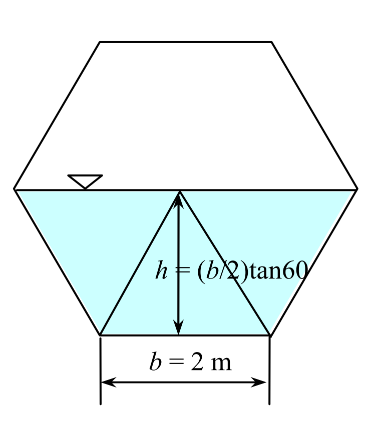
***Assumptions*** The flow is uniform.

***Analysis*** (*a*) The flow area is determined from geometric considerations to be

$$A_c = \frac{(b+2b)}{2}\frac{b}{2}\tan 60° = \frac{(2+2\times 2)\ \text{m}}{2}\frac{2\ \text{m}}{2}\tan 60° = 5.196\ \text{m}^2$$

Then the average velocity becomes

$$V = \frac{\dot{V}}{A_c} = \frac{45\ \text{m}^3/\text{s}}{5.196\ \text{m}^2} = \mathbf{8.66\ m/s}$$




(*b*) When calculating the Froude number, the hydraulic depth should be used rather than the maximum depth or the hydraulic radius. For a non-rectangular channel, hydraulic depth is defined as the ratio of the flow area to top width,

$$y = y_h = \frac{A_c}{\text{Top width}} = \frac{A_c}{2b} = \frac{5.196\ \text{m}}{2\times 2\ \text{m}} = 1.299\ \text{m}$$

Then the Froude number becomes

$$\text{Fr} = \frac{V}{\sqrt{gy}} = \frac{8.66\ \text{m/s}}{\sqrt{(9.81\ \text{m/s}^2)(1.299\ \text{m})}} = 2.43$$

which is greater than 1. Therefore, the flow is **supercritical**.

***Discussion*** The analysis is approximate since the edge effects are significant here compared to a wide rectangular channel, and thus the results should be interpreted accordingly.

---

**13-40**

**Solution** Water flows uniformly through a half-full hexagon channel. For a given flow rate, the average velocity and whether the flow is subcritical or supercritical are to be determined.

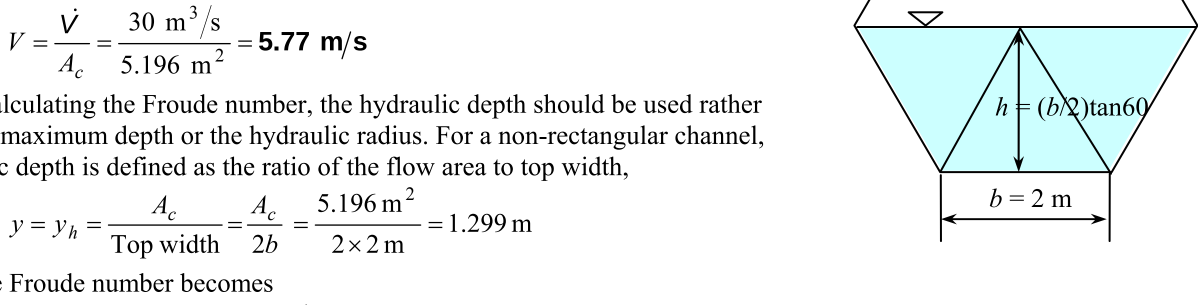
***Assumptions*** The flow is uniform.

***Analysis*** The flow area is determined from geometric considerations to be

$$A_c = \frac{(b+2b)}{2}\frac{b}{2}\tan 60° = \frac{(2+2\times 2)\text{ m}}{2}\frac{2\text{ m}}{2}\tan 60° = 5.196\ \text{m}^2$$

Then the average velocity becomes

$$V = \frac{\dot{V}}{A_c} = \frac{30\ \text{m}^3/\text{s}}{5.196\ \text{m}^2} = \mathbf{5.77\ m/s}$$




When calculating the Froude number, the hydraulic depth should be used rather than the maximum depth or the hydraulic radius. For a non-rectangular channel, hydraulic depth is defined as the ratio of the flow area to top width,

$$y = y_h = \frac{A_c}{\text{Top width}} = \frac{A_c}{2b} = \frac{5.196\,\text{m}^2}{2\times 2\,\text{m}} = 1.299\,\text{m}$$

Then the Froude number becomes

$$\text{Fr} = \frac{V}{\sqrt{gy}} = \frac{5.77\text{ m/s}}{\sqrt{(9.81\,\text{m/s}^2)(1.299\,\text{m})}} = 1.62$$

which is greater than 1. Therefore, the flow is **supercritical**.

***Discussion*** The analysis is approximate since the edge effects are significant here compared to a wide rectangular channel, and thus the results should be interpreted accordingly.

## Uniform Flow and Best Hydraulic Cross Sections

**13-41C**

**Solution** We are to discuss when flow in an open channel is uniform, and how it remains uniform.

***Analysis*** Flow in a channel is called *uniform flow* if the **flow depth (and thus the average flow velocity) remains constant**. **The flow remains uniform as long as the slope, cross-section, and the surface roughness of the channel remain unchanged**.

***Discussion*** Uniform flow in open-channel flow is somewhat analogous to fully developed pipe flow in internal flow.

**13-42C**

**Solution** We are to discuss how flow depth changes when the bottom slope is increased.

***Analysis*** The flow depth **decreases** when the bottom slope is increased.

***Discussion*** You can think of it in simple terms this way: As the slope increases, the liquid flows faster, and faster flow requires lower depth.

**13-43C**

**Solution** We are to examine a claim that head loss can be determined by multiplying bottom slope by channel length.

***Analysis*** **Yes, the claim is correct**. The head loss in uniform flow is $h_L = S_0 L$ since the head loss must equal elevation loss.

***Discussion*** In uniform flow, frictional head losses are exactly balanced by elevation loss, which is directly proportional to bottom slope.

---

**13-44C**

**Solution** We are to discuss the constants and coefficients in the Manning equation.

***Analysis*** The value of the factor $a$ in SI units is $\boldsymbol{a}$ **= 1 m**$^{1/3}$**/s**. Combining the relations $C = \sqrt{8g/f}$ and $C = \frac{a}{n} R_h^{1/6}$ and solving them for $n$ gives the desired relation to be $\boxed{n = \frac{a}{\sqrt{8g/f}} R_h^{1/6}}$. In practice, $n$ is usually determined experimentally.

***Discussion*** The value of $n$ varies greatly with surface roughness.

---

**13-45C**

**Solution** It is to be shown that for uniform critical flow, the general critical slope relation $S_c = \frac{gn^2 y_c}{a^2 R_h^{4/3}}$ reduces to $S_c = \frac{gn^2}{a^2 y_c^{1/3}}$ for film flow with $b >> y_c$.

***Analysis*** For critical flow, the flow depth is $y = y_c$. For film flow, the hydraulic radius is $R_h = y = y_c$. Substituting into the critical slope relation gives the desired result, $\boxed{S_c = \frac{gn^2 y_c}{a^2 R_h^{4/3}} = \frac{gn^2 y_c}{a^2 y_c^{4/3}} = \frac{gn^2}{a^2 y_c^{1/3}}}$.

***Discussion*** The reduced equation is valid for film flow only – be careful not to apply it to channels of other shapes.

---

**13-46C**

**Solution** We are to determine which cross section is better – one with a small or large hydraulic radius.

***Analysis*** The best hydraulic cross-section for an open channel is the one with the **maximum hydraulic radius**, or equivalently, the one with the minimum wetted perimeter for a specified cross-sectional area.

***Discussion*** Frictional losses occur at the wetted perimeter walls of the channel, so it makes sense to minimize the wetted perimeter in order to minimize the frictional losses.

---

**13-47C**

**Solution** We are to determine which cross section shape is best for an open channel.

***Analysis*** The best hydraulic cross-section for an open channel is a (*a*) **circular** one.

***Discussion*** Circular channels are often more difficult to construct, however, so they are often not used in practice.

---

**13-48C**

**Solution** We are to determine the best hydraulic cross section for a rectangular channel.

***Analysis*** The best hydraulic cross section for a rectangular channel is one whose fluid height is (*a*) **half the channel width**.

***Discussion*** It turns out that for this case, the wetted perimeter, and thus the frictional losses, are smallest.

---

**13-49C**

**Solution** We are to determine the best hydraulic cross section for a trapezoidal channel.

***Analysis*** The best hydraulic cross section for a trapezoidal channel of base width *b* is (*a*) one for which **the length of the side edge of the flow section is *b***.

***Discussion*** It turns out that for this case, the wetted perimeter, and thus the frictional losses, are smallest.

---

**13-50C**

**Solution** We are to determine how the flow rate changes when the Manning coefficient doubles.

***Analysis*** The flow rate in uniform flow is given as $\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2}$, and thus the flow rate is inversely proportional to the Manning coefficient. Therefore, if the Manning coefficient doubles as a result of some algae growth on surfaces while the flow cross section remains constant, the flow rate will (*d*) **decrease by half**.

***Discussion*** In an actual case, the cross section may also change due to flow depth changes as well.

---

**13-51**

**Solution** The flow of water in a trapezoidal finished-concrete channel is considered. For a given flow depth and bottom slope, the flow rate is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

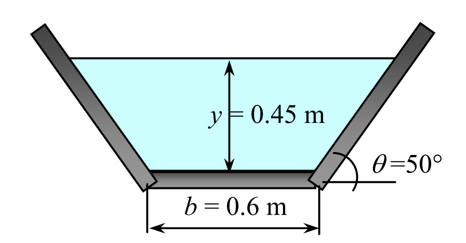
***Properties*** Manning coefficient for an open channel of finished concrete is $n = 0.012$ (Table 13-1).

***Analysis*** The flow area, wetted perimeter, and hydraulic radius of the channel are

$$A_c = y\left(b + \frac{y}{\tan\theta}\right) = (0.45\text{ m})\left(0.60\text{ m} + \frac{0.45\text{ m}}{\tan 50°}\right) = 0.3724\ \text{m}^2$$

$$p = b + \frac{2y}{\sin\theta} = 0.6\text{ m} + \frac{2(0.45\text{ m})}{\sin 50°} = 1.775\text{ m}$$

$$R_h = \frac{A_c}{p} = \frac{0.3724\text{ m}^2}{1.775\text{ m}} = 0.2096\ \text{m}$$




Bottom slope of the channel is

$$S_0 = \tan 0.4° = 0.006981$$

Then the flow rate can be determined from Manning's equation to be

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} = \frac{1\text{ m}^{1/3}/s}{0.012}(0.3724\text{ m}^2)(0.2096\text{ m})^{2/3}(0.006981)^{1/2} = \mathbf{0.915\ m^3/s}$$

***Discussion*** Note that the flow rate in a given channel is a strong function of the bottom slope.

---

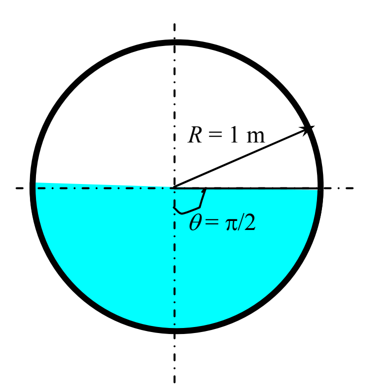
## 13-52

**Solution** Water flows uniformly half-full in a circular finished-concrete channel. For a given bottom slope, the flow rate is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Properties*** Manning coefficient for an open channel of finished concrete is $n$ = 0.012 (Table 13-1).

***Analysis*** The flow area, wetted perimeter, and hydraulic radius of the channel are




$$A_c = \frac{\pi R^2}{2} = \frac{\pi(1\,\text{m})^2}{2} = 1.571\,\text{m}^2$$

$$p = \frac{2\pi R}{2} = \frac{2\pi(1\,\text{m})}{2} = 3.142\,\text{m}$$

$$R_h = \frac{A_c}{P} = \frac{\pi R^2/2}{\pi R} = \frac{R}{2} = \frac{1\,\text{m}}{2} = 0.50\ \text{m}$$



Then the flow rate can be determined from Manning's equation to be

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} = \frac{1\,\text{m}^{1/3}/s}{0.012}(1.571\,\text{m}^2)(0.50\,\text{m})^{2/3}(1.5/1000)^{1/2} = \mathbf{3.19\ m^3/s}$$

***Discussion*** Note that the flow rate in a given channel is a strong function of the bottom slope.

---

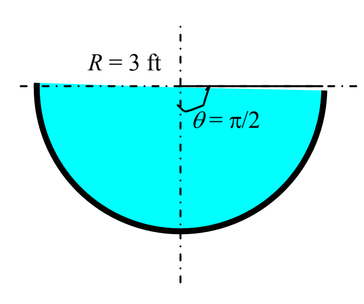
## 13-53E

**Solution** Water is to be transported uniformly in a full semi-circular unfinished-concrete channel. For a specified flow rate, the elevation difference across the channel is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Properties*** Manning coefficient for an open channel of unfinished concrete is $n$ = 0.014 (Table 13-1).

***Analysis*** The flow area, wetted perimeter, and hydraulic radius of the channel are




$$A_c = \frac{\pi R^2}{2} = \frac{\pi(3\,\text{ft})^2}{2} = 14.14\,\text{ft}^2$$

$$p = \frac{2\pi R}{2} = \frac{2\pi(3\,\text{ft})}{2} = 9.425\,\text{ft}$$

$$R_h = \frac{A_c}{P} = \frac{\pi R^2/2}{\pi R} = \frac{R}{2} = \frac{3\,\text{ft}}{2} = 1.50\ \text{ft}$$

Substituting the given quantities into Manning's equation,

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} \rightarrow \quad 150\,\text{ft}^3/\text{s} = \frac{1.486\,\text{ft}^{1/3}/s}{0.014}(14.14\,\text{ft}^2)(1.50\,\text{ft})^{2/3} S_0^{1/2}$$

It gives the slope to be $S_0$ = 0.005817. Therefore, the *elevation difference* $\Delta z$ across a pipe length of $L$ = 1 mile = 5280 ft must be

$$\Delta z = S_0 L = 0.005817(5280\,\text{ft}) = \mathbf{30.7\ ft}$$

***Discussion*** Note that when transporting water through a region of fixed elevation drop, the only way to increase the flow rate is to use a channel with a larger cross-section.

---

**13-54**

**Solution** Water is to be transported uniformly in a trapezoidal asphalt-lined channel. For a specified flow rate, the required elevation drop per km channel length is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

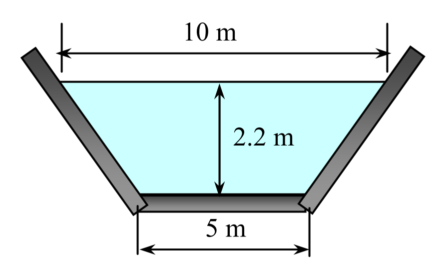
***Properties*** Manning coefficient for an asphalt-lined open channel is $n$ = 0.016 (Table 13-1).

***Analysis*** The flow area, wetted perimeter, and hydraulic radius of the channel are

$$A_c = \frac{10\text{ m} + 5\text{ m}}{2}(2.2\text{ m}) = 16.5\text{ m}^2$$

$$p = (5\text{ m}) + 2\sqrt{(2.2\text{ m})^2 + (2.5\text{ m})^2} = 11.66\text{ m}$$

$$R_h = \frac{A_c}{p} = \frac{16.5\text{ m}^2}{11.66\text{ m}} = 1.415\text{ m}$$




Substituting the given quantities into Manning's equation,

$$\dot{V} = \frac{a}{n}A_c R_h^{2/3} S_0^{1/2} \quad \rightarrow \quad 120\text{ m}^3/\text{s} = \frac{1\text{ m}^{1/3}/s}{0.016}(16.5\text{ m}^2)(1.415\text{ m})^{2/3} S_0^{1/2}$$

It gives the slope to be $S_0$ = 0.008524. Therefore, the *elevation drop* $\Delta z$ across a pipe length of $L$ = 1 km must be

$$\Delta z = S_0 L = 0.008524(1000\text{ m}) = \mathbf{8.52\ m}$$

***Discussion*** Note that when transporting water through a region of fixed elevation drop, the only way to increase the flow rate is to use a channel with a larger cross-section.

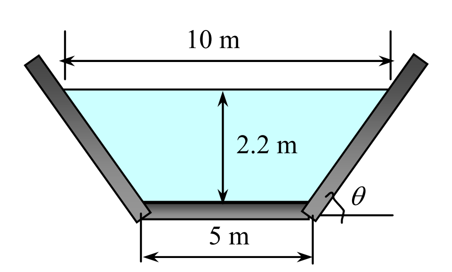
**13-55**

**Solution** The flow of water through the trapezoidal asphalt-lined channel in the previous problem is reconsidered. The maximum flow rate corresponding to a given maximum channel height is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Analysis*** We denote the flow conditions in the previous problem by subscript 1 and the conditions for the maximum case in this problem by subscript 2. Using the Manning's equation $\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2}$ and noting that the Manning coefficient and the channel slope remain constant, the flow rate in case 2 can be expressed in terms of flow rate in case 1 as

$$\frac{\dot{V}_2}{\dot{V}_1} = \frac{(a/n) A_{c2} R_{h2}^{2/3}}{(a/n) A_{c1} R_{h1}^{2/3}} \quad \rightarrow \quad \dot{V}_2 = \frac{A_{c2}}{A_{c1}} \left( \frac{R_{h2}}{R_{h1}} \right)^{2/3} \dot{V}_1$$



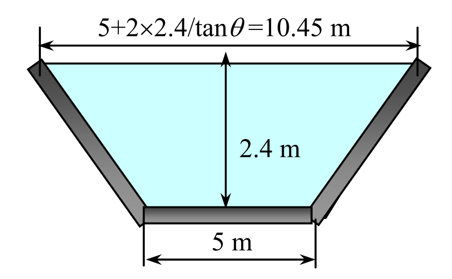

The trapezoid angle is $\tan\theta = 2.2/2.5 = 0.88 \rightarrow \theta = 2.2/2.5 = 41.34°$.

From geometric considerations,

$$A_{c1} = \frac{10\text{ m} + 5\text{ m}}{2}(2.2\text{ m}) = 16.5\text{ m}^2$$

$$p_1 = (5\text{ m}) + 2\sqrt{(2.2\text{ m})^2 + (2.5\text{ m})^2} = 11.66\text{ m}$$

$$R_{h1} = \frac{A_{c1}}{p_1} = \frac{16.5\text{ m}^2}{11.66\text{ m}} = 1.415\text{ m}$$




and

$$A_{c2} = \frac{10.45\text{ m} + 5\text{ m}}{2}(2.4\text{ m}) = 18.54\text{ m}^2$$

$$p_2 = (5\text{ m}) + 2\sqrt{(2.4\text{ m})^2 + (5.45/2\text{ m})^2} = 12.26\text{ m}$$

$$R_{h2} = \frac{A_{c2}}{p_2} = \frac{18.54\text{ m}^2}{12.26\text{ m}} = 1.512\text{ m}$$

Substituting,

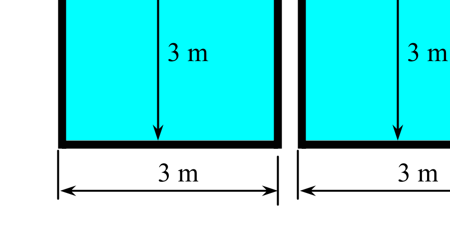
$$\dot{V}_2 = \frac{A_{c2}}{A_{c1}} \left( \frac{R_{h2}}{R_{h1}} \right)^{2/3} \dot{V}_1 = \frac{18.54\text{ m}^2}{16.5\text{ m}^2} \left( \frac{1.512\text{ m}}{1.415\text{ m}} \right)^{2/3} (120\text{ m}^3/\text{s}) = \mathbf{141\ m^3/s}$$

***Discussion*** Note that a 9% increase in flow depth results in an 18% increase in flow rate.

**13-56**

**Solution** The flow of water through two identical channels with square flow sections is considered. The percent increase in flow rate as a result of combining the two channels while the flow depth remains constant is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.




***Analysis*** We denote the flow conditions for two separate channels by subscript 1 and the conditions for the combined wide channel by subscript 2. Using the Manning's equation $\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2}$ and noting that the Manning coefficient, channel slope, and the flow area $A_c$ remain constant, the flow rate in case 2 can be expressed in terms of flow rate in case 1 as

$$\frac{\dot{V}_2}{\dot{V}_1} = \frac{(a/n)A_{c2}R_{h2}^{2/3}}{(a/n)A_{c1}R_{h1}^{2/3}} = \left(\frac{R_{h2}}{R_{h1}}\right)^{2/3} = \left(\frac{A_{c2}/p_2}{A_{c1}/p_1}\right)^{2/3} = \left(\frac{p_1}{p_2}\right)^{2/3}$$

where $p$ is the wetted perimeter. Substituting,

$$\frac{\dot{V}_2}{\dot{V}_1} = \left(\frac{p_2}{p_2}\right)^{2/3} = \left(\frac{6\times 3\text{ m}}{4\times 3\text{ m}}\right)^{2/3} = \left(\frac{3}{2}\right)^{2/3} = \mathbf{1.31} \qquad \textbf{(31\% increase)}$$

***Discussion*** This is a very significant increase, and shows the importance of eliminating unnecessary surfaces in flow systems, including pipe flow.

---

**13-57**

**Solution** The flow of water in a trapezoidal channel made of unfinished-concrete is considered. For given flow rate and bottom slope, the flow depth is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

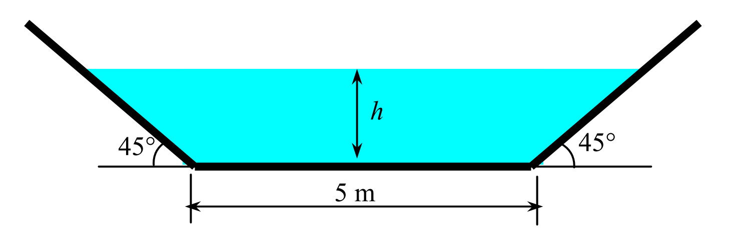
***Properties*** Manning coefficient for an open channel of unfinished concrete is $n$ = 0.014 (Table 13-1).

***Analysis*** From geometric considerations, the flow area, wetted perimeter, and hydraulic radius are

$$A_c = \frac{5\text{ m}+5\text{ m}+2h}{2}h = (5+h)h$$

$$p = (5\text{ m}) + 2h/\sin 45° = 5 + 2.828h$$

$$R_h = \frac{A_c}{p} = \frac{(5+h)h}{5+2h/\sin 45°}$$




Substituting the given quantities into Manning's equation,

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} \quad\rightarrow\quad 25\text{ m}^3/\text{s} = \frac{1\text{ m}^{1/3}/s}{0.014}(5+h)h\left(\frac{(5+h)h}{5+2h/\sin 45°}\right)^{2/3}(\tan 1°)^{1/2}$$

It gives the flow depth to be $h$ = **0.685 m**.

***Discussion*** Non-linear equations frequently arise in the solution of open channel flow problems. They are best handled by equation solvers such as EES.

---

**13-58**

**Solution** The flow of water in a weedy excavated trapezoidal channel is considered. For given flow rate and bottom slope, the flow depth is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

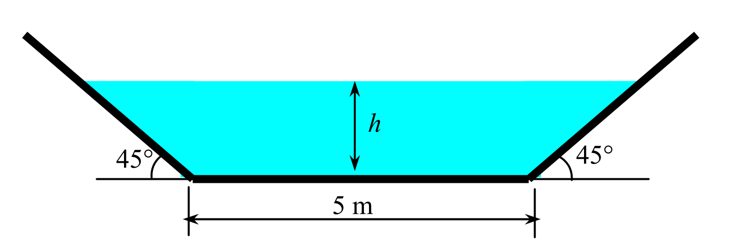
***Properties*** Manning coefficient for the channel is given to be $n = 0.030$.

***Analysis*** From geometric considerations, the flow area, wetted perimeter, and hydraulic radius are

$$A_c = \frac{5\text{ m} + 5\text{ m} + 2h}{2}h = (5+h)h$$

$$p = (5\text{ m}) + 2h/\sin 45° = 5 + 2.828h$$

$$R_h = \frac{A_c}{p} = \frac{(5+h)h}{5 + 2h/\sin 45°}$$




Substituting the given quantities into Manning's equation,

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} \quad \to \quad 25\text{ m}^3/\text{s} = \frac{1\text{ m}^{1/3}/s}{0.030}(5+h)h\left(\frac{(5+h)h}{5+2h/\sin 45°}\right)^{2/3}(\tan 1°)^{1/2}$$

It gives the flow depth to be $y$ = **1.07 m**.

***Discussion*** Note that as the Manning coefficient increases because of the increased surface roughness of the channel, the flow depth required to maintain the same flow rate also increases.

---

**13-59**

**Solution** The flow of water in a V-shaped cast iron channel is considered. For a given flow depth and bottom slope, the flow rate is to be determined.

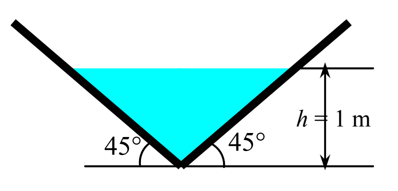
***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Properties*** Manning coefficient for an open channel of cast iron is $n = 0.013$ (Table 13-1).

***Analysis*** The flow area, wetted perimeter, and hydraulic radius of the channel are

$$A_c = \frac{2h \times h}{2} = h^2 = (1\text{ m})^2 = 1\text{ m}^2 \qquad p = 2h/\sin\theta = 2(1\text{ m})/\sin 45° = 2\sqrt{2}\text{ m}$$

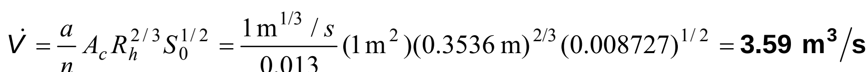
$$R_h = \frac{A_c}{p} = \frac{1\text{ m}^2}{2\sqrt{2}\text{ m}} = 0.3536\text{ m}$$




The bottom slope of the channel is $S_0 = \tan 0.5° = 0.008727$.



Then the flow rate can be determined from Manning's equation to be



$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} = \frac{1\text{ m}^{1/3}/s}{0.013}(1\text{ m}^2)(0.3536\text{ m})^{2/3}(0.008727)^{1/2} = \mathbf{3.59\ m^3/s}$$

***Discussion*** Note that the flow rate in a given channel is a strong function of the bottom slope.

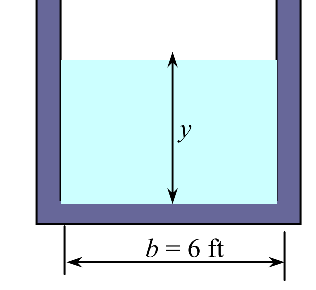
---

**13-60E**

**Solution** The flow of water in a rectangular cast iron channel is considered. For given flow rate and bottom slope, the flow depth is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** The bottom slope is constant. **3** The roughness coefficient is constant.

***Properties*** Manning coefficient for a cast iron open channel is $n$ = 0.013 (Table 13-1).




***Analysis*** From the geometry, the flow area, wetted perimeter, and hydraulic radius are

$$A_c = by = (6\text{ ft})y = 6y \qquad p = (6\text{ ft}) + 2y = 6 + 2y \qquad R_h = \frac{A_c}{p} = \frac{6y}{6+2y}$$

The channel bottom slope is $S_0$ = 1.5/1000 = 0.0015.
Substituting the given quantities into Manning's equation,

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} \quad \rightarrow \quad 70\text{ ft}^3/\text{s} = \frac{1.486\text{ ft}^{1/3}/s}{0.013}(6y)\left(\frac{6y}{6+2y}\right)^{2/3}(0.0015)^{1/2}$$

Solution of the above equation gives the flow depth to be $h$ = **2.24 ft**.

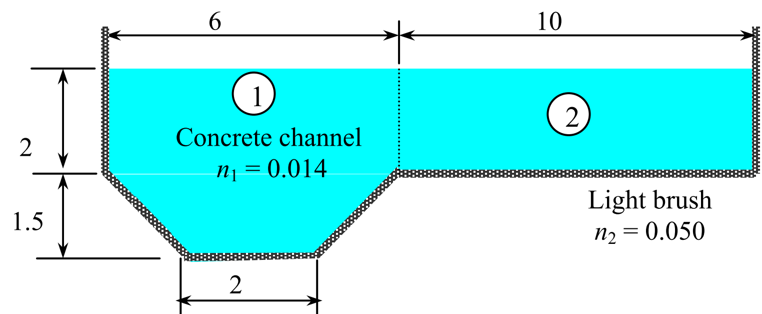
***Discussion*** Non-linear equations frequently arise in the solution of open channel flow problems. They are best handled by equation solvers such as EES.

---

**13-61**

**Solution** Water is flowing through a channel with nonuniform surface properties. The flow rate and the effective Manning coefficient are to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** The bottom slope is constant. **3** The Manning coefficients do not vary along the channel.




***Analysis*** The channel involves two parts with different roughness, and thus it is appropriate to divide the channel into two subsections. The flow rate for each subsection can be determined from the Manning equation, and the total flow rate can be determined by adding them up.
The flow area, perimeter, and hydraulic radius for each subsection and the entire channel are:

Subsection 1: $A_{c1} = 18\text{ m}^2, \quad p_1 = 9\text{ m}, \quad R_{h1} = \frac{A_{c1}}{p_1} = \frac{18\text{ m}^2}{9\text{ m}} = 2.00\text{ m}$

Subsection 2: $A_{c2} = 20\text{ m}^2, \quad p_2 = 12\text{ m}, \quad R_{h2} = \frac{A_{c2}}{p_2} = \frac{20\text{ m}^2}{12\text{ m}} = 1.67\text{ m}$

Entire channel: $A_c = 38\text{ m}^2, \quad p = 21\text{ m}, \quad R_h = \frac{A_c}{p} = \frac{38\text{ m}^2}{21\text{ m}} = 1.81\text{ m}$

Applying the Manning equation to each subsection, the total flow rate through the channel is determined to be

$$\dot{V} = \dot{V}_1 + \dot{V}_2 = \frac{a}{n_1} A_1 R_1^{2/3} S_0^{1/2} + \frac{a}{n_1} A_1 R_1^{2/3} S_0^{1/2} = \left(1\text{ m}^{1/3}/\text{s}\right)\left(\frac{(18\text{ m}^2)\ (2\text{ m})^{2/3}}{0.014} + \frac{(20\text{ m}^2)\ (1.67\text{ m})^{2/3}}{0.05}\right)(0.002)^{1/2} = \mathbf{116\ m^3/s}$$

Knowing the total flow rate, the effective Manning coefficient for the entire channel can be determined from the Manning equation to be

$$n_{\text{eff}} = \frac{aA_c R_h^{2/3} S_0^{1/2}}{\dot{V}} = \frac{(1\text{ m}^{1/3}/\text{s})(38\text{ m}^2)(1.81\text{ m})^{2/3}(0.002)^{1/2}}{116\text{ m}^3/\text{s}} = \mathbf{0.0217}$$

***Discussion*** The effective Manning coefficient $n_{\text{eff}}$ lies between the two $n$ values as expected. The weighted average of the Manning coefficient of the channel is $n_{\text{ave}}=(n_1p_1+ n_2p_2)/p$ = 0.035, which is quite different than $n_{\text{eff}}$. Therefore, using a weighted average Manning coefficient for the entire channel may be tempting, but it would not be accurate.

---

**13-62**

**Solution** Waters flows in a partially filled circular channel made of finished concrete. For a given flow depth and bottom slope, the flow rate is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Properties*** Manning coefficient for an open channel made of finished concrete is $n = 0.012$ (Table 13-1).

***Analysis*** From geometric considerations,

$$\cos\theta = \frac{R-y}{R} = \frac{0.5-0.25}{0.5} = 0.5 \quad \rightarrow \quad \theta = 60° = 60\frac{2\pi}{360} = \frac{\pi}{3}$$

$$A_c = R^2(\theta - \sin\theta\cos\theta) = (0.5\text{ m})^2[\pi/3 - \sin(\pi/3)\cos(\pi/3)] = 0.1535\text{ m}^2$$

$$R_h = \frac{A_c}{p} = \frac{\theta - \sin\theta\cos\theta}{2\theta}R = \frac{\pi/3 - \sin(\pi/3)\cos(\pi/3)}{2\pi/3}(0.5\text{ m}) = 0.1466\text{ m}$$

$R = 0.5$ m

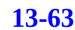
$\theta$

$y = 0.25$ m

Then the flow rate can be determined from Manning's equation to be

$$\dot{V} = \frac{a}{n}A_c R_h^{2/3} S_0^{1/2} = \frac{1\text{ m}^{1/3}/s}{0.012}(0.1535\text{ m}^2)(0.1466\text{ m})^{2/3}(0.002)^{1/2} = \mathbf{0.159\ m^3/s}$$

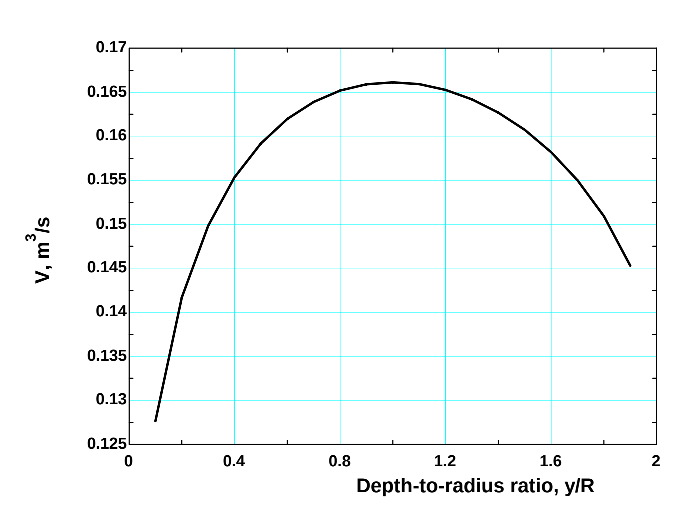
***Discussion*** Note that the flow rate in a given channel is a strong function of the bottom slope.


**13-63** 

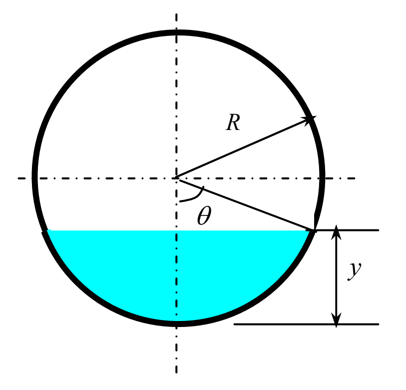
**Solution** The previous problem is reconsidered. By varying the flow depth-to-radius ratio from 0.1 to 1.9 for a fixed value of flow area, it is the to be shown that the best hydraulic cross section occurs when the circular channel is half-full, and the results are to be plotted.

***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
a=1
n=0.012
s=0.002
Ac=0.1536 “Flow area kept constant”
ratio=y/R “This ratio is varied from 0.1 to 1.9”
bdeg=arcsin((R-y)/R)
tetadeg=90-bdeg
teta=tetadeg*2*pi/360
Ac=R^2*(teta-sin(tetadeg)*cos(tetadeg))
p=2*teta*R
Rh=Ac/p
Vdot=(a/n)*Ac*Rh^(2/3)*SQRT(s)
```




| Depth-to-radius ratio, $y/R$ | Channel radius, $R$, m | Flow rate, $\dot{V}$, m$^3$/s |
|---|---|---|
| 0.1 | 1.617 | 0.1276 |
| 0.2 | 0.969 | 0.1417 |
| 0.3 | 0.721 | 0.1498 |
| 0.4 | 0.586 | 0.1553 |
| 0.5 | 0.500 | 0.1592 |
| 0.6 | 0.440 | 0.1620 |
| 0.7 | 0.396 | 0.1639 |
| 0.8 | 0.362 | 0.1652 |
| 0.9 | 0.335 | 0.1659 |
| **1.0** | 0.313 | **0.1661** |
| 1.1 | 0.295 | 0.1659 |
| 1.2 | 0.279 | 0.1653 |
| 1.3 | 0.267 | 0.1642 |
| 1.4 | 0.256 | 0.1627 |
| 1.5 | 0.247 | 0.1607 |
| 1.6 | 0.239 | 0.1582 |
| 1.7 | 0.232 | 0.1550 |
| 1.8 | 0.227 | 0.1509 |
| 1.9 | 0.223 | 0.1453 |




***Discussion*** The depth-to-radius ratio of $y/R = 1$ corresponds to a half-full circular channel, and it is clear from the table and the chart that, for a fixed flow area, the flow rate becomes maximum when the channel is half-full.

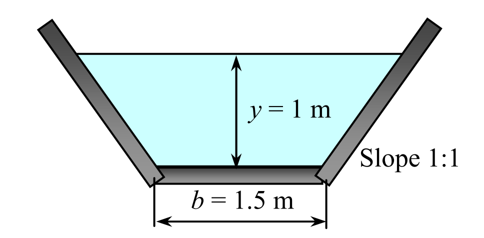
## 13-64

**Solution** Water is to be transported uniformly in a clean-earth trapezoidal channel. For a specified flow rate, the required elevation drop per km channel length is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Properties*** Manning coefficient for the clean-earth lined open channel is $n = 0.022$ (Table 13-1).

***Analysis*** The flow area, wetted perimeter, and hydraulic radius of the channel are




$$A_c = \frac{(1.5+1.5+2)\ \text{m}}{2}(1\,\text{m}) = 2.5\,\text{m}^2$$

$$p = (1.5\,\text{m}) + 2\sqrt{(1\,\text{m})^2 + (1\,\text{m})^2} = 4.328\,\text{m}$$

$$R_h = \frac{A_c}{p} = \frac{2.5\,\text{m}^2}{4.328\,\text{m}} = 0.5776\,\text{m}$$

Substituting the given quantities into Manning's equation,

$$\dot{V} = \frac{a}{n}A_c R_h^{2/3} S_0^{1/2} \quad\rightarrow\quad 8\,\text{m}^3/\text{s} = \frac{1\,\text{m}^{1/3}/s}{0.022}(2.5\,\text{m}^2)(0.5776\,\text{m})^{2/3} S_0^{1/2}$$

It gives the slope to be $S_0 = 0.0103$. Therefore, the *elevation drop* $\Delta z$ across a pipe length of $L = 1$ km must be

$$\Delta z = S_0 L = 0.0103(1000\,\text{m}) = \mathbf{10.3\ m}$$

***Discussion*** Note that when transporting water through a region of fixed elevation drop, the only way to increase the flow rate is to use a channel with a larger cross-section.

---

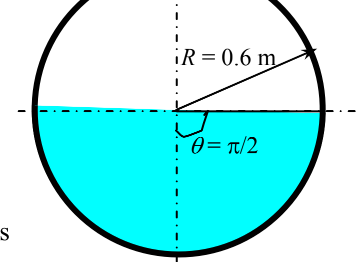
## 13-65

**Solution** A water draining system consists of three circular channels, two of which draining into the third one. If all channels are to run half-full, the diameter of the third channel is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel. **4** Losses at the junction are negligible.

***Properties*** The Manning coefficient for asphalt lined open channels is $n = 0.016$ (Table 13-1).

***Analysis*** The flow area, wetted perimeter, and hydraulic radius of the two pipes upstream are




$$A_c = \frac{\pi R^2}{2} = \frac{\pi(0.6\,\text{m})^2}{2} = 0.5655\,\text{m}^2 \qquad p = \frac{2\pi R}{2} = \frac{2\pi(0.6\,\text{m})}{2} = 1.885\,\text{m}$$

$$R_h = \frac{A_c}{P} = \frac{\pi R^2/2}{\pi R} = \frac{R}{2} = \frac{0.6\,\text{m}}{2} = 0.30\ \text{m}$$

Then the flow rate through the 2 pipes becomes, from Manning's equation,

$$\dot{V} = 2\frac{a}{n}A_c R_h^{2/3} S_0^{1/2} = 2\frac{1\,\text{m}^{1/3}/s}{0.016}(0.5655\,\text{m}^2)(0.30\,\text{m})^{2/3}(0.0015)^{1/2} = 1.227\,\text{m}^3/\text{s}$$

The third channel is half-full, and the flow rate through it remains the same. Noting that the flow area is $\pi R^2/2$ and the hydraulic radius is $R/2$, we have

$$1.227\,\text{m}^3/\text{s} = \frac{1\,\text{m}^{1/3}/s}{0.016}(\pi R^2/2\,\text{m}^2)(R/2\,\text{m})^{2/3}(0.0015)^{1/2}$$

Solving for $R$ gives $R = 0.778$ m. Therefore, the diameter of the third channel is $D_3$ **= 1.56 m** .

***Discussion*** Note that if the channel diameter were larger, the channel would have been less than half full.

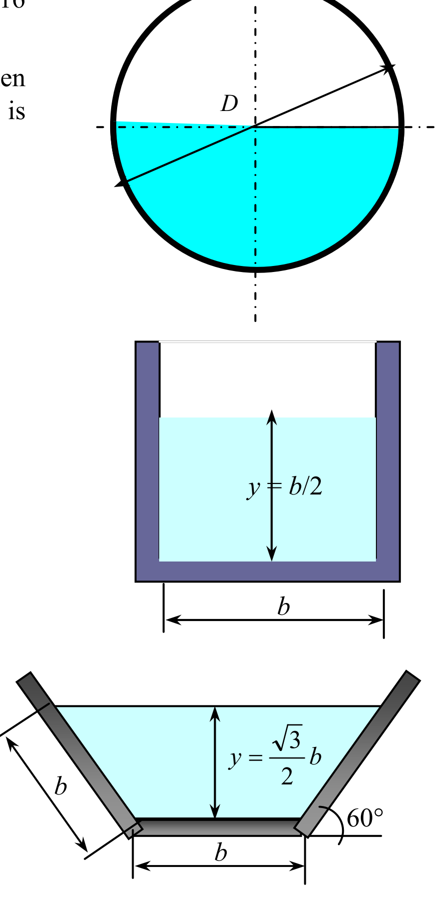
---

**13-66**

**Solution** Water is transported in an asphalt lined open channel at a specified rate. The dimensions of the best cross-section for various geometric shapes are to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Properties*** The Manning coefficient for asphalt lined open channels is $n = 0.016$ (Table 13-1).




***Analysis*** (*a*) Circular channel of Diameter *D*: Best cross-section occurs when the channel is half-full, and thus the flow area is $\pi D^2/8$ and the hydraulic radius is $D/4$. Then from Manning's equation, $\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2}$,

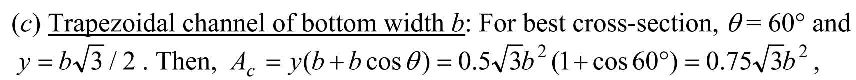
$$4\ \text{m}^3/\text{s} = \frac{1\ \text{m}^{1/3}/s}{0.016}(\pi D^2/8\ \text{m}^2)(D/4\ \text{m})^{2/3}(0.0015)^{1/2}$$

which gives $D$ **= 2.42 m.**

(*b*) Rectangular channel of bottom width *b*: For best cross-section, $y = b/2$. Then $A_c = yb = b^2/2$ and $R_h = b/4$. From the Manning equation,

$$4\ \text{m}^3/\text{s} = \frac{1\ \text{m}^{1/3}/s}{0.016}(b^2/2\ \text{m}^2)(b/4\ \text{m})^{2/3}(0.0015)^{1/2}$$

which gives $b$ **= 2.21 m,** and $y = b/2$ **= 1.11 m.**



(*c*) Trapezoidal channel of bottom width *b*: For best cross-section, $\theta = 60°$ and $y = b\sqrt{3}/2$. Then, $A_c = y(b + b\cos\theta) = 0.5\sqrt{3}b^2(1+\cos 60°) = 0.75\sqrt{3}b^2$,

$p = 3b$, $R_h = \frac{y}{2} = \frac{\sqrt{3}}{4}b$. From the Manning equation,

$$4\ \text{m}^3/\text{s} = \frac{1\ \text{m}^{1/3}/s}{0.016}(0.75\sqrt{3}b^2\ \text{m}^2)(\sqrt{3}b/4\ \text{m})^{2/3}(0.0015)^{1/2}$$

which gives $b$ **= 1.35 m,** and $y$ **= 1.17 m** and $\theta$ **= 60°.**

***Discussion*** The perimeters for the circular, rectangular, and trapezoidal channels are 3.80 m, 4.42 m, and 4.05 m, respectively. Therefore, the circular cross-section has the smallest perimeter.

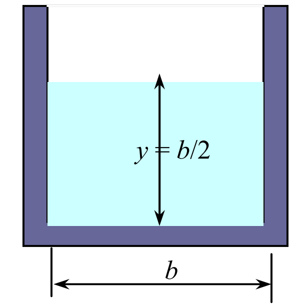
**13-67E**

**Solution** Water is to be transported in a rectangular channel at a specified rate. The dimensions for the best cross-section if the channel is made of unfinished and finished concrete are to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Properties*** The Manning coefficient is $n$ = 0.012 and $n$ = 0.014 for finished and unfinished concrete, respectively (Table 13-1).

***Analysis*** For best cross-section of a rectangular cross-section, $y = b/2$. Then $A_c = yb = b^2/2$ and $R_h = b/4$. The flow rate is determined from the Manning equation, $\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2}$,




($a$) Finished concrete, $n$ = 0.012:

$$800 \text{ ft}^3/\text{s} = \frac{1.486 \text{ ft}^{1/3}/s}{0.012}(b^2/2 \text{ ft}^2)(b/4 \text{ ft})^{2/3}(0.0005)^{1/2}$$

It gives $b$ **= 15.4 ft,** and $y = b/2$ = **7.68 ft**

($b$) Unfinished concrete, $n$ = 0.014:

$$800 \text{ ft}^3/\text{s} = \frac{1.486 \text{ ft}^{1/3}/s}{0.014}(b^2/2 \text{ ft}^2)(b/4 \text{ ft})^{2/3}(0.0005)^{1/2}$$

It gives $b$ **= 16.3 ft,** and $y = b/2$ = **8.13 ft**

***Discussion*** Note that channels with rough surfaces require a larger cross-section to transport the same amount of water.

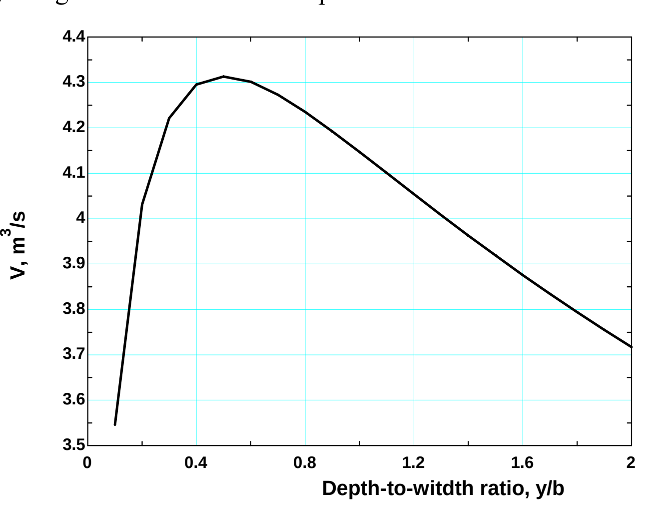


**13-68** 

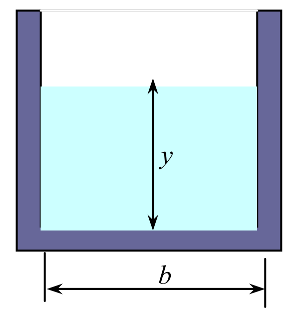
**Solution** Uniform flow in an asphalt-lined rectangular channel is considered. By varying the depth-to-width ratio from 0.1 to 2 in increments of 0.1 for a fixed value of flow area, it is the to be shown that the best hydraulic cross section occurs when $y/b$ = 0.5, and the results are to be plotted.

***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
a=1
n=0.016 "Manning coefficient"
s=0.003 "Bottom slope is constant"
Ac=2 "Flow area remains constant at 2 m2"
Ratio=y/b
Ac=b*y
p=b+2*y
Rh=Ac/p "Hydraulic radius"
Vdot=(a/n)*Ac*Rh^(2/3)*SQRT(s) "Volume flow rate"
```




| Depth-to-width ratio, $y/b$ | Channel width, $b$, m | Flow rate, $\dot{V}$, $m^3/s$ |
|---|---|---|
| 0.1 | 4.47 | 3.546 |
| 0.2 | 3.16 | 4.031 |
| 0.3 | 2.58 | 4.221 |
| 0.4 | 2.24 | 4.295 |
| **0.5** | 2.00 | **4.313** |
| 0.6 | 1.83 | 4.301 |
| 0.7 | 1.69 | 4.273 |
| 0.8 | 1.58 | 4.235 |
| 0.9 | 1.49 | 4.192 |
| 1.0 | 1.41 | 4.147 |
| 1.1 | 1.35 | 4.101 |
| 1.2 | 1.29 | 4.054 |
| 1.3 | 1.24 | 4.008 |
| 1.4 | 1.20 | 3.963 |
| 1.5 | 1.15 | 3.919 |
| 1.6 | 1.12 | 3.876 |
| 1.7 | 1.08 | 3.834 |
| 1.8 | 1.05 | 3.794 |
| 1.9 | 1.03 | 3.755 |
| 2.0 | 1.00 | 3.717 |




***Discussion*** It is clear from the table and the chart that the depth-to-width ratio of $y/b$ = 0.5 corresponds to the best cross-section for an open channel of rectangular cross-section.

## Gradually and Rapidly Varied Flows and Hydraulic Jump

### 13-69C

**Solution** We are to discuss the difference between uniform and nonuniform (varied) flow.

***Analysis*** Both uniform and varied flows are steady, and thus neither involves any change with time at a specified location. In *uniform flow*, **the flow depth *y* and the flow velocity *V* remain constant** whereas in *nonuniform* or *varied flow*, **the flow depth and velocity vary in the streamwise direction of the flow**. In *uniform flow*, **the slope of the energy line is equal to the slope of the bottom surface**. Therefore, the friction slope equals the bottom slope, $S_f = S_0$. In *varied flow*, however, **these slopes are different**.

***Discussion*** Varied flows are further classified into gradually varied flow (GVF) and rapidly varied flow (RVF).

---

### 13-70C

**Solution** We are to discuss the differences between GVF and RVF.

***Analysis*** *Gradually varied flow* (GVF) is characterized by **gradual variations in flow depth and velocity** (small slopes and no abrupt changes) and **a free surface that always remains smooth** (no discontinuities or zigzags). *Rapidly varied flow* (RVF) involves **rapid changes in flow depth and velocity**. A change in the bottom slope or cross-section of a channel or an obstruction on the path of flow may cause the uniform flow in a channel to become gradually or rapidly varied flow. **Analytical relations for the profile of the free surface can be obtained in GVF, but this is not the case for RVF because of the intense agitation**.

***Discussion*** In many situations, the shape of the free surface must be solved numerically, even for GVF.

---

### 13-71C

**Solution** We are to analyze a claim that wall shear is negligible in RVF but important in GVF.

***Analysis*** **Yes, we agree with this claim**. Rapidly varied flows occur over a short section of the channel with relatively small surface area, and thus frictional losses associated with wall shear are negligible compared with losses due to intense agitation and turbulence. Losses in GVF, on the other hand, are primarily due to frictional effects along the channel, and should be considered.

***Discussion*** There is somewhat of an analogy here with internal flows. In long pipe sections with entrance lengths and/or gradually changing pipe diameter, wall shear is important. However, in short sections of piping with rapid change of diameter or a blockage or turn, etc (minor loss), friction along the wall is typically negligible compared to other losses.

---

### 13-72C

**Solution** We are to analyze what happens to flow depth in a horizontal rectangular channel during subcritical flow.

***Analysis*** The flow depth *y* must (*c*) ***decrease* in the flow direction**.

***Discussion*** Since the flow is subcritical, there is no possibility of a hydraulic jump.

---

### 13-73C

**Solution** We are to analyze what happens to flow depth in a sloped rectangular channel during subcritical flow.

***Analysis*** The flow depth *y* must (*a*) ***increase* in the flow direction**.

***Discussion*** Since the flow is subcritical, there is no possibility of a hydraulic jump.

---

**13-74C**

**Solution** We are to analyze what happens to flow depth in a horizontal rectangular channel during supercritical flow.

***Analysis*** The flow depth $y$ ($a$) ***increases* in the flow direction**.

***Discussion*** Since the flow is supercritical, this increase in flow depth may occur via a hydraulic jump.

---

**13-75C**

**Solution** We are to analyze what happens to flow depth in a sloped rectangular channel during subcritical flow.

***Analysis*** The flow depth $y$ ($c$) ***decreases* in the flow direction**.

***Discussion*** Since the flow is subcritical, there is no possibility of a hydraulic jump.

---

**13-76C**

**Solution** We are to analyze what happens to flow depth in an upward-sloped rectangular channel during supercritical flow.

***Analysis*** The flow depth $y$ ($a$) ***increases* in the flow direction**.

***Discussion*** Since the flow is supercritical, this increase in flow depth may occur via a hydraulic jump.

---

**13-77C**

**Solution** We are to determine if it is possible for subcritical flow to undergo a hydraulic jump.

***Analysis*** **No. It is impossible for subcritical flow to undergo a hydraulic jump**. Such a process would require the head loss $h_L$ to become negative, which is impossible. It would correspond to negative entropy generation, which would be a violation of the second law of thermodynamics. Therefore, the upstream flow must be supercritical ($Fr_1 > 1$) for a hydraulic jump to occur.

***Discussion*** This is analogous to **normal shock waves** in gases – the only way a shock wave can occur is if the flow upstream of the shock wave is supersonic with $Ma_1 > 1$ (analogous to supercritical in open-channel flow with $Fr_1 > 1$).

---

**13-78C**

**Solution** We are to define the energy dissipation ratio for a hydraulic jump and discuss why a hydraulic jump is sometimes used to dissipate energy.

***Analysis*** Hydraulic jumps are often designed in conjunction with stilling basins and spillways of dams in order to waste as much of the mechanical energy as possible **to minimize the mechanical energy of the fluid and thus its potential to cause damage**. In such cases, a measure of performance of a hydraulic jump is the *energy dissipation ratio*, which is the **fraction of energy dissipated** through a hydraulic jump, defined as

$$\boxed{\text{Dissipation ratio} = \frac{h_L}{E_{s1}} = \frac{h_L}{y_1 + V_1^2/(2g)} = \frac{h_L}{y_1\left(1+\text{Fr}_1^2/2\right)}}.$$

***Discussion*** Since the head loss is always positive, the dissipation ratio is also always positive.

---

## 13-79

**Solution** Water is flowing in an open channel uniformly. It is to be determined whether the channel slope is mild, critical, or steep for this flow.

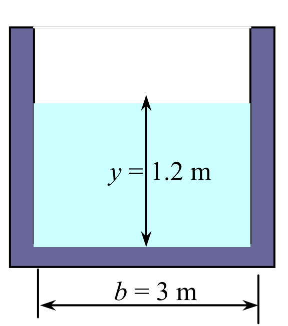
***Assumptions*** **1** The flow is steady and uniform. **2** The bottom slope is constant. **3** The roughness of the wetted surface of the channel and thus the friction coefficient are constant.

***Properties*** The Manning coefficient for an open channel with finished concrete surfaces is $n = 0.012$ (Table 13-1).

***Analysis*** The cross-sectional area, perimeter, and hydraulic radius are

$$A_c = yb = (1.2\text{ m})(3\text{ m}) = 3.6\text{ m}^2 \qquad p = b + 2y = 3\text{ m} + 2(1.2\text{ m}) = 5.4\text{ m}$$

$$R_h = \frac{A_c}{p} = \frac{3.6\text{ m}^2}{5.4\text{ m}} = 0.6667\text{ m}$$




The flow rate is determined from the Manning equation to be

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} = \frac{1\text{ m}^{1/3}/\text{s}}{0.012}(3.6\text{ m}^2)(0.6667\text{ m})^{2/3}(0.002)^{1/2} = 10.2\text{ m}^3/\text{s}$$

Noting that the flow is uniform, the specified flow rate is the normal depth and thus $y = y_n = 1.2$ m. The critical depth for this flow is

$$y_c = \left(\frac{\dot{V}^2}{g b^2}\right)^{1/3} = \left(\frac{(10.2\text{ m}^3/\text{s})^2}{(9.81\text{ m/s}^2)(3\text{ m})^2}\right)^{1/3} = 1.06\text{ m}$$

This channel at these flow conditions is classified as **mild** since $y > y_c$, and the flow is subcritical.

***Discussion*** If the flow depth were smaller than 1.06 m, the channel slope would be said to be *steep*. Therefore, the bottom slope alone is not sufficient to classify a downhill channel as being mild, critical, or steep.



## 13-80

**Solution** Water is flowing in an wide brick open channel uniformly. The range of flow depth for which the channel can be classified as “steep” is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** The bottom slope is constant. **3** The roughness of the wetted surface of the channel and thus the friction coefficient are constant.

***Properties*** The Manning coefficient for a brick open channel is $n = 0.015$ (Table 13-1).

***Analysis*** The slope of the channel is $S_0 = \tan\alpha = \tan 0.4° = 0.006981$.

The hydraulic radius for a wide channel is equal to the flow depth, $R_h = y$. Now assume the flow in the channel to be critical, The channel flow in this case would be critical slope $S_c$, and the flow depth would be the critical flow depth, which is determined from

$$S_c = \frac{g n^2}{a^2 y_c^{1/3}} \quad \rightarrow \quad y_c = \left(\frac{g n^2}{a^2 S_c}\right)^3$$

Substituting,

$$y_c = \left(\frac{g n^2}{a^2 S_c}\right)^3 = \left(\frac{(9.81\text{ m/s}^2)(0.015)^2}{(1\text{ m}^{1/3}/s)^2(0.006981)}\right)^3 = 0.0316\text{ m}$$

Therefore, this channel can be classified as *steep* for uniform flow depths less than $y_c$, i.e., $\boldsymbol{y < 0.0316}$ **m**.

***Discussion*** Note that two channels of the same slope can be classified as differently (one mild and the other steep) if they have different roughness and thus different values of $n$.

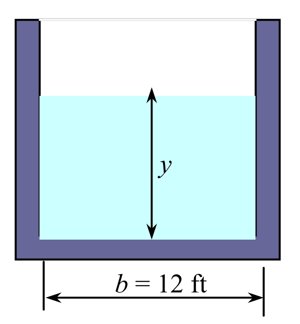
## 13-81E

**Solution** Water is flowing in a rectangular open channel with a specified bottom slope at a specified flow rate. It is to be determined whether the slope of this channel should be classified as mild, critical, or steep. The surface profile is also to be classified for a specified flow depth of 2 m.

***Assumptions*** **1** The flow is steady and uniform. **2** The bottom slope is constant. **3** The roughness of the wetted surface of the channel and thus the friction coefficient are constant.

***Properties*** The Manning coefficient of a channel with unfinished concrete surfaces is $n$ = 0.014 (Table 13-1).

***Analysis*** The cross-sectional area, perimeter, and hydraulic radius are




$$A_c = yb = y(12\text{ ft}) = 12y\text{ ft}^2 \qquad p = b + 2y = 12\text{ ft} + 2y = 12 + 2y\text{ ft}$$

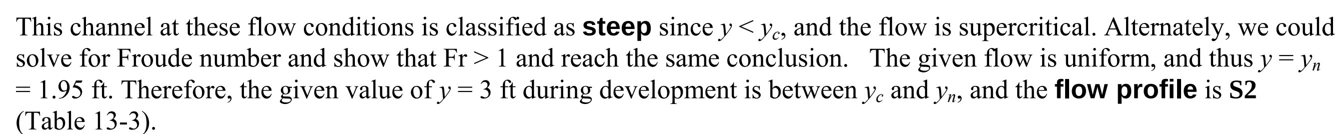
$$R_h = \frac{A_c}{p} = \frac{12y\text{ ft}^2}{12 + 2\text{y ft}} = 0.6667\text{ m}$$

Substituting the known quantities into the Manning equation,

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} \quad \rightarrow \quad 300\text{ ft}^3/\text{s} = \frac{1.486\text{ ft}^{1/3}/s}{0.014}(12y)\left(\frac{12y}{12+2y}\right)^{2/3}(\tan 0.5°)^{1/2}$$

Solving for the flow depth $y$ gives $y$ = 1.95 ft. The critical depth for this flow is

$$y_c = \frac{\dot{V}^2}{gA_c^2} = \frac{(300\text{ ft}^3/\text{s})^2}{(32.2\text{ ft/s}^2)(12\text{ ft}\times 1.95\text{ ft})^2} = 5.10\text{ ft}$$



This channel at these flow conditions is classified as **steep** since $y < y_c$, and the flow is supercritical. Alternately, we could solve for Froude number and show that Fr > 1 and reach the same conclusion. The given flow is uniform, and thus $y = y_n$ = 1.95 ft. Therefore, the given value of $y$ = 3 ft during development is between $y_c$ and $y_n$, and the **flow profile** is **S2** (Table 13-3).

***Discussion*** If the flow depth were larger than 5.19 ft, the channel slope would be said to be *mild*. Therefore, the bottom slope alone is not sufficient to classify a downhill channel as being mild, critical, or steep.

---



## 13-82

**Solution** Water is flowing in a V-shaped open channel with a specified bottom slope at a specified rate. It is to be determined whether the slope of this channel should be classified as mild, critical, or steep.

***Assumptions*** **1** The flow is steady and uniform. **2** The bottom slope is constant. **3** The roughness of the wetted surface of the channel and thus the friction coefficient are constant.

***Properties*** The Manning coefficient for a cast iron channel is $n$ = 0.013 (Table 13-1).

***Analysis*** From geometric considerations, the cross-sectional area, perimeter, and hydraulic radius are

$$A_c = y(2y)/2 = y^2 \qquad p = 2\sqrt{y^2 + y^2} = 2\sqrt{2}y \qquad R_h = \frac{A_c}{p} = \frac{y^2}{2\sqrt{2}\text{y}} = \frac{y}{2\sqrt{2}}$$

Substituting the known quantities into the Manning equation,

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} \quad \rightarrow 3\text{ m}^3/\text{s} = \frac{1\text{ m}^{1/3}/\text{s}}{0.013}(y^2)\left(\frac{\text{y}}{2\sqrt{2}}\right)^{2/3}(0.002)^{1/2}$$

Solving for the flow depth $y$ gives $y$ = 1.23 m. The critical depth for this flow is

$$y_c = \frac{\dot{V}^2}{gA_c^2} = \frac{(3\text{ m}^3/\text{s})^2}{(9.81\text{ m/s}^2)(1.23\text{ m})^2} = 0.61\text{ m}$$

This channel at these flow conditions is classified as **mild** since $y > y_c$, and the flow is subcritical.

***Discussion*** If the flow depth were smaller than 0.61 m, the channel slope would be said to be *steep*. Therefore, the bottom slope alone is not sufficient to classify a downhill channel as being mild, critical, or steep.

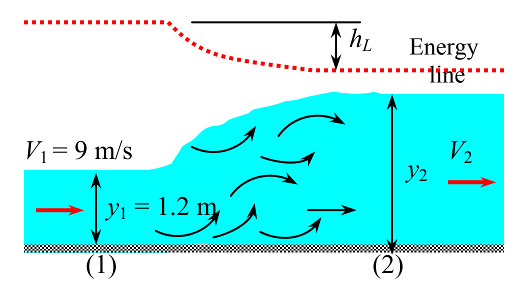
---

**13-83** [*Also solved using EES on enclosed DVD*]

**Solution** Water at a specified depth and velocity undergoes a hydraulic jump. The depth and Froude number after the jump, the head loss and dissipation ratio, and dissipated mechanical power are to be determined.

***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible. **3** The channel is horizontal.

***Properties*** The density of water is 1000 kg/m³.




***Analysis*** (*a*) The Froude number before the hydraulic jump is

$$Fr_1 = \frac{V_1}{\sqrt{gy_1}} = \frac{9\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(1.2\text{ m})}} = 2.62$$

which is greater than 1. Therefore, the flow is supercritical before the jump. The flow depth, velocity, and Froude number after the jump are

$$y_2 = 0.5y_1\left(-1+\sqrt{1+8Fr_1^2}\right) = 0.5(1.2\text{ m})\left(-1+\sqrt{1+8\times 2.62^2}\right) = \mathbf{3.89\text{ m}}$$

$$V_2 = \frac{y_1}{y_2}V_1 = \frac{1.2\text{ m}}{3.89\text{ m}}(9\text{ m/s}) = 2.78\text{ m/s}$$

$$Fr_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{2.78\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(3.89\text{ m})}} = \mathbf{0.449}$$

(*b*) The head loss is determined from the energy equation to be

$$h_L = y_1 - y_2 + \frac{V_1^2 - V_2^2}{2g} = (1.2\text{ m}) - (3.89\text{ m}) + \frac{(9\text{ m/s})^2 - (2.78\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = \mathbf{1.05\text{ m}}$$

The specific energy of water before the jump and the dissipation ratio are

$$E_{s1} = y_1 + \frac{V_1^2}{2g} = (1.2\text{ m}) + \frac{(9\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 5.33\text{ m}$$

$$\text{Dissipation ratio} = \frac{h_L}{E_{s1}} = \frac{1.04\text{ m}}{5.33\text{ m}} = \mathbf{0.195}$$

Therefore, 19.5% of the available head (or mechanical energy) of the liquid is wasted (converted to thermal energy) as a result of frictional effects during this hydraulic jump.

(*c*) The mass flow rate of water is

$$\dot{m} = \rho\dot{V} = \rho b y_1 V_1 = (1000\text{ kg/m}^3)(1.2\text{ m})(8\text{ m})(9\text{ m/s}) = 86{,}400\text{ kg/s}$$

Then the dissipated mechanical power becomes

$$\dot{E}_{\text{dissipated}} = \dot{m}gh_L = (86{,}400\text{ kg/s})(9.81\text{ m/s}^2)(1.04\text{ m})\left(\frac{1\text{ N}}{1\text{ kg}\cdot\text{m/s}^2}\right) = 881{,}000\text{ Nm/s} = \mathbf{881\text{ kW}}$$

***Discussion*** The results show that the hydraulic jump is a highly dissipative process, wasting 881 kW of power production potential in this case. That is, if the water is routed to a hydraulic turbine instead of being released from the sluice gate, up to 881 kW of power could be produced.

**13-84**

**Solution** Water at a specified depth and velocity undergoes a hydraulic jump. The head loss associated with this process is to be determined.

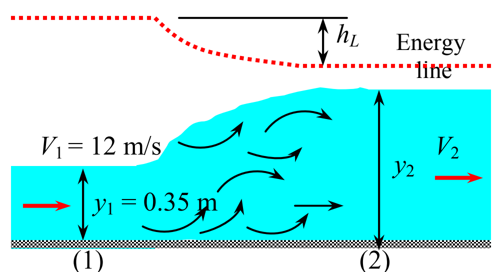
***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible. **3** The channel is horizontal.

***Analysis*** The Froude number before the hydraulic jump is $\mathrm{Fr}_1 = \dfrac{V_1}{\sqrt{gy_1}} = \dfrac{12\ \mathrm{m/s}}{\sqrt{(9.81\ \mathrm{m/s^2})(0.35\ \mathrm{m})}} = 6.476$, which is greater than 1. Therefore, the flow is indeed supercritical before the jump. The flow depth, velocity, and Froude number after the jump are

$$y_2 = 0.5y_1\left(-1+\sqrt{1+8\mathrm{Fr}_1^2}\right) = 0.5(0.35\ \mathrm{m})\left(-1+\sqrt{1+8\times 6.476^2}\right) = 3.035\ \mathrm{m}$$

$$V_2 = \frac{y_1}{y_2}V_1 = \frac{0.35\ \mathrm{m}}{3.035\ \mathrm{m}}(12\ \mathrm{m/s}) = 1.384\ \mathrm{m/s}$$

$$\mathrm{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{1.384\ \mathrm{m/s}}{\sqrt{(9.81\ \mathrm{m/s^2})(3.035\ \mathrm{m})}} = 0.2536$$




The head loss is determined from the energy equation to be

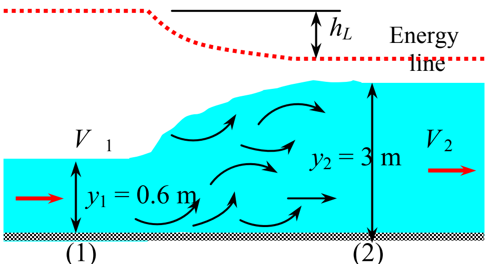
$$h_L = y_1 - y_2 + \frac{V_1^2 - V_2^2}{2g} = (0.35\ \mathrm{m}) - (3.035\ \mathrm{m}) + \frac{(12\ \mathrm{m/s})^2 - (1.384\ \mathrm{m/s})^2}{2(9.81\ \mathrm{m/s^2})} = \mathbf{4.56\ m}$$

***Discussion*** The results show that the hydraulic jump is a highly dissipative process, wasting 4.56 m of head in the process.

**13-85**

**Solution** The increase in flow depth during a hydraulic jump is given. The velocities and Froude numbers before and after the jump, and the energy dissipation ratio are to be determined.

***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible. **3** The channel is horizontal.




***Analysis*** The Froude number before the jump is determined from

$$y_2 = 0.5y_1\left(-1+\sqrt{1+8\mathrm{Fr}_1^2}\right) \rightarrow \quad 3\text{ m} = 0.5\times(0.6\text{ m})\left(-1+\sqrt{1+8\mathrm{Fr}^2}\right)$$

which gives $\mathrm{Fr}_1 =$ **3.873.** Then,

$$V_1 = \mathrm{Fr}_1\sqrt{gy_1} = 3.873\sqrt{(9.81\text{ m/s}^2)(0.6\text{ m})} = \mathbf{9.40\ m/s}$$

$$V_2 = \frac{y_1}{y_2}V_1 = \frac{0.6\text{ m}}{3\text{ m}}(9.40\text{ m/s}) = \mathbf{1.88\ m/s}$$

$$\mathrm{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{1.88\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(3\text{ m})}} = \mathbf{0.347}$$

The head loss is determined from the energy equation to be

$$h_L = y_1 - y_2 + \frac{V_1^2 - V_2^2}{2g} = (0.6\text{ m}) - (3\text{ m}) + \frac{(9.40\text{ m/s})^2 - (1.88\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = \mathbf{1.92\ m}$$

The specific energy of water before the jump and the dissipation ratio are

$$E_{s1} = y_1 + \frac{V_1^2}{2g} = (0.6\text{ m}) + \frac{(9.40\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 5.10\text{ m}$$

$$\text{Dissipation ratio} = \frac{h_L}{E_{s1}} = \frac{1.92\text{ m}}{5.10\text{ m}} = \mathbf{0.376}$$

Therefore, 37.6% of the available head (or mechanical energy) of water is wasted (converted to thermal energy) as a result of frictional effects during this hydraulic jump.

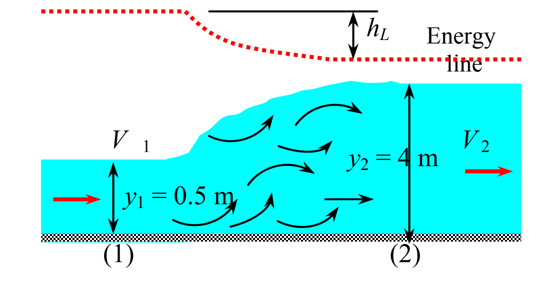
***Discussion*** The results show that the hydraulic jump is a highly dissipative process, wasting over one-third of the available head.

**13-86**

**Solution** Water flowing in a wide channel at a specified depth and flow rate undergoes a hydraulic jump. The mechanical power wasted during this process is to be determined.

***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible. **3** The channel is horizontal.

***Properties*** The density of water is 1000 kg/m$^3$.




***Analysis*** Average velocities before and after the jump are

$$V_1 = \frac{70\ \text{m}^3/\text{s}}{(10\ \text{m})(0.5\ \text{m})} = 14\ \text{m/s}$$

$$V_2 = \frac{70\ \text{m}^3/\text{s}}{(10\ \text{m})(4\ \text{m})} = 1.75\ \text{m/s}$$

The head loss is determined from the energy equation to be

$$h_L = y_1 - y_2 + \frac{V_1^2 - V_2^2}{2g} = (0.5\ \text{m}) - (4\ \text{m}) + \frac{(14\ \text{m/s})^2 - (1.75\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)} = 6.33\ \text{m}$$

The mass flow rate of water is

$$\dot{m} = \rho \dot{V} = (1000\ \text{kg/m}^3)(70\ \text{m}^3/\text{s}) = 70{,}000\ \text{kg/s}$$

Then the dissipated mechanical power becomes

$$\dot{E}_{\text{dissipated}} = \dot{m} g h_L = (70{,}000\ \text{kg/s})(9.81\ \text{m/s}^2)(6.33\ \text{m})\left(\frac{1\ \text{kN}}{1000\ \text{kg}\cdot\text{m/s}^2}\right) = 4350\ \text{kNm/s} = \mathbf{4.35\ MW}$$

***Discussion*** The results show that the hydraulic jump is a highly dissipative process, wasting 4.35 MW of power production potential in this case.

**13-87**

**Solution** The flow depth and average velocity of water after a hydraulic jump are measured. The flow depth and velocity before the jump as well as the fraction of mechanical energy dissipated are to be determined.

***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible. **3** The channel is horizontal.

***Analysis*** The Froude number after the hydraulic jump is

$$\mathrm{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{3\ \mathrm{m/s}}{\sqrt{(9.81\ \mathrm{m/s^2})(2\ \mathrm{m})}} = 0.6773$$

It can be shown that the subscripts in the relation

$y_2 = 0.5y_1\left(-1+\sqrt{1+8\mathrm{Fr}_1^2}\right)$ are interchangeable. Thus,

$$y_1 = 0.5y_2\left(-1+\sqrt{1+8\mathrm{Fr}_2^2}\right) = 0.5(2\ \mathrm{m})\left(-1+\sqrt{1+8\times 0.6773^2}\right) = 1.16\ \mathrm{m}$$

$$V_1 = \frac{y_2}{y_1}V_2 = \frac{2\ \mathrm{m}}{1.161\ \mathrm{m}}(3\ \mathrm{m/s}) = 5.17\ \mathrm{m/s}$$

The Froude number before the jump is

$$\mathrm{Fr}_1 = \frac{V_1}{\sqrt{gy_1}} = \frac{5.17\ \mathrm{m/s}}{\sqrt{(9.81\ \mathrm{m/s^2})(1.16\ \mathrm{m})}} = 1.53$$

which is greater than 1. Therefore, the flow is indeed supercritical before the jump. The head loss is determined from the energy equation to be

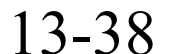
$$h_L = y_1 - y_2 + \frac{V_1^2 - V_2^2}{2g} = (1.16\ \mathrm{m}) - (2\ \mathrm{m}) + \frac{(5.17\ \mathrm{m/s})^2 - (3\ \mathrm{m/s})^2}{2(9.81\ \mathrm{m/s^2})} = 0.0636\ \mathrm{m}$$

The specific energy of water before the jump and the dissipation ratio is

$$E_{s1} = y_1 + \frac{V_1^2}{2g} = (1.16\ \mathrm{m}) + \frac{(5.17\ \mathrm{m/s})^2}{2(9.81\ \mathrm{m/s^2})} = 2.52\ \mathrm{m}$$

$$\text{Dissipation ratio} = \frac{h_L}{E_{s1}} = \frac{0.0636\ \mathrm{m}}{2.52\ \mathrm{m}} = \mathbf{0.025}$$

***Discussion*** Note that this is a "mild" hydraulic jump, and only 2.5% of the available energy is wasted.



## 13-88E

**Solution** Water at a specified depth and velocity undergoes a hydraulic jump, and dissipates a known fraction of its energy. The flow depth, velocity, and Froude number after the jump and the head loss associated with the jump are to be determined.

***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible. **3** The channel is horizontal.

***Analysis*** The Froude number before the hydraulic jump is

$$\mathrm{Fr}_1 = \frac{V_1}{\sqrt{gy_1}} = \frac{40 \text{ ft/s}}{\sqrt{(32.2 \text{ m/s}^2)(2 \text{ ft})}} = 4.984$$

$h_L$ Energy line

$V_1 = 40$ ft/s

$y_1 = 2$ ft

$y_2$

$V_2$

(1) (2)

which is greater than 1. Therefore, the flow is indeed supercritical before the jump. The flow depth, velocity, and Froude number after the jump are

$$y_2 = 0.5y_1\left(-1+\sqrt{1+8\mathrm{Fr}_1^2}\right) = 0.5(2 \text{ ft})\left(-1+\sqrt{1+8\times 4.984^2}\right) = \mathbf{13.1\ ft}$$

$$V_2 = \frac{y_1}{y_2}V_1 = \frac{2 \text{ ft}}{13.1 \text{ ft}}(40 \text{ ft/s}) = \mathbf{6.09\ ft/s}$$

$$\mathrm{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{6.091 \text{ ft/s}}{\sqrt{(32.2 \text{ ft/s}^2)(13.13 \text{ m})}} = \mathbf{0.296}$$

The head loss is determined from the energy equation to be

$$h_L = y_1 - y_2 + \frac{V_1^2 - V_2^2}{2g} = (2 \text{ ft}) - (13.1 \text{ ft}) + \frac{(40 \text{ ft/s})^2 - (6.09 \text{ ft/s})^2}{2(32.2 \text{ ft/s}^2)} = \mathbf{13.2\ ft}$$

***Discussion*** The results show that the hydraulic jump is a highly dissipative process, wasting 13.2 ft of head in the process.

---

## Flow Control and Measurement in Channels

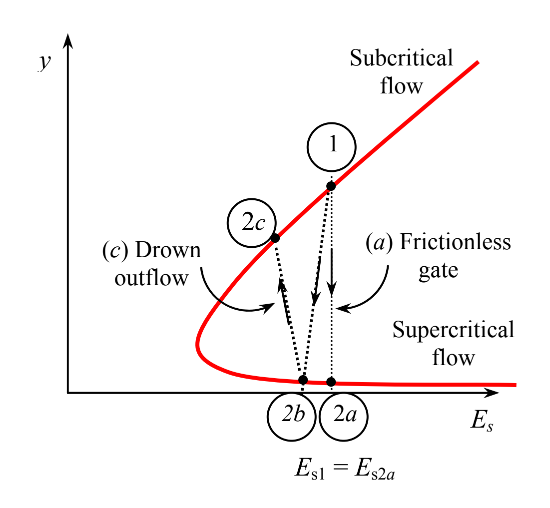
---

## 13-89C

**Solution** We are to draw a flow depth-specific energy diagram for several types of flow.

***Analysis*** On the figure, diagram 1-2a is for frictionless gate, **1-2b is for sluice gate with free outflow**, and **1-2b-2c is for sluice gate with drown outflow, including the hydraulic jump back to subcritical flow**.

***Discussion*** A plot of flow depth as a function of specific energy, as shown here, is quite useful in the analysis of varied open-channel flow because the states upstream and downstream of a change must jump between the two branches.




---

**13-90C**

**Solution** We are to define the discharge coefficient for sluice gates, and discuss some typical values.

***Analysis*** For sluice gates, the *discharge coefficient* $C_d$ is defined as **the ratio of the actual velocity through the gate to the maximum velocity** as determined by the Bernoulli equation for the idealized frictionless flow case. **For ideal flow, $C_d$ = 1**. Typical values of $C_d$ for sluice gates with free outflow are in the range of **0.55 to 0.60**.

***Discussion*** Actual values of the discharge coefficient must be less than one or else the second law would be violated.

**13-91C**

**Solution** We are to discuss how flow rate is measured with a broad-crested weir.

***Analysis*** The operation of broad crested weir is based on **blocking the flow in the channel with a rectangular block, and establishing critical flow over the block**. Then the flow rate is determined by measuring flow depths.

***Discussion*** This technique is quite obtrusive, but requires no special measuring equipment or probes.

**13-92C**

**Solution** We are to analyze whether the free surface of flow over a bump will increase, decrease, or remain constant.

***Analysis*** In the case of *subcritical flow*, the **flow depth *y* will decrease during flow over the bump**.

***Discussion*** This may be contrary to our intuition at first, but if we think in terms of increasing velocity and decreasing pressure over the bump (a Bernoulli type of analysis), it makes sense that the surface will decrease over the bump.

**13-93C**

**Solution** We are to analyze what happens in subcritical flow over a bump when the bump height increases.

***Analysis*** When the specific energy reaches its minimum value, **the flow is critical, and the flow at this point is said to be choked**. If the bump height is increased even further, **the flow remains critical and thus choked. The flow will not become supercritical**.

***Discussion*** This is somewhat analogous to compressible flow in a converging nozzle – the flow cannot become supersonic at the nozzle exit unless there is a diverging section of the nozzle downstream of the throat.

**13-94C**

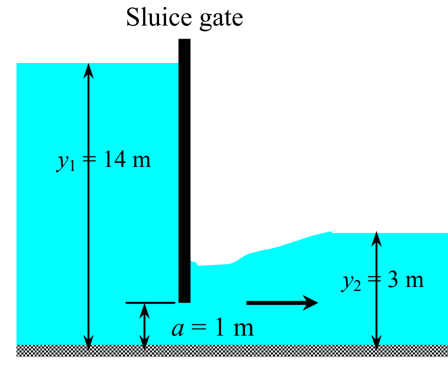
**Solution** We are to define and classify sharp-crested weirs.

***Analysis*** A *sharp-crested weir* is a **vertical plate placed in a channel that forces the fluid to flow through an opening to measure the flow rate**. They are **characterized by the shape of the opening**. For example, a weir with a triangular opening is referred to as a triangular weir.

***Discussion*** Similar to the broad-crested weir, this type of flow measurement is quite obtrusive, but requires no special measuring equipment or probes.

## 13-95

**Solution** Water is released from a reservoir through a sluice gate into an open channel. For specified flow depths, the rate of discharge is to be determined.




***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The depth ratio $y_1/a$ and the contraction coefficient $y_2/a$ are

$$\frac{y_1}{a}=\frac{14\text{ m}}{1\text{ m}}=14 \quad \text{and} \quad \frac{y_2}{a}=\frac{3\text{ m}}{1\text{ m}}=3$$

The corresponding discharge coefficient is determined from Fig. 13-38 to be $C_d$ = 0.59. Then the discharge rate becomes

$$\dot{V}=C_d ba\sqrt{2gy_1}=0.59(5\text{ m})(1\text{ m})\sqrt{2(9.81\text{ m/s}^2)(14\text{ m})}=\mathbf{48.9\ m^3/s}$$

***Discussion*** Discharge coefficient is the same as free flow because of small depth ratio after the gate. So, the flow rate would not change if it were not drowned.

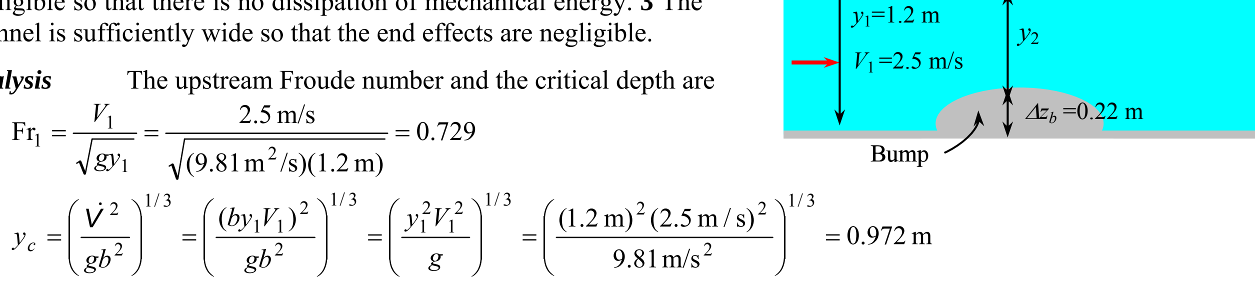
---

## 13-96

**Solution** Water flowing in a horizontal open channel encounters a bump. It will be determined if the flow over the bump is choked.

***Assumptions*** **1** The flow is steady. **2** Frictional effects are negligible so that there is no dissipation of mechanical energy. **3** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The upstream Froude number and the critical depth are




$$\text{Fr}_1=\frac{V_1}{\sqrt{gy_1}}=\frac{2.5\text{ m/s}}{\sqrt{(9.81\text{ m}^2\text{/s})(1.2\text{ m})}}=0.729$$

$$y_c=\left(\frac{\dot{V}^2}{gb^2}\right)^{1/3}=\left(\frac{(by_1V_1)^2}{gb^2}\right)^{1/3}=\left(\frac{y_1^2V_1^2}{g}\right)^{1/3}=\left(\frac{(1.2\text{ m})^2(2.5\text{ m/s})^2}{9.81\text{ m/s}^2}\right)^{1/3}=0.972\text{ m}$$

The flow is subcritical since Fr < 1, and the flow depth decreases over the bump. The upstream, over the bump, and critical specific energies are

$$E_{s1}=y_1+\frac{V_1^2}{2g}=(1.2\text{ m})+\frac{(2.5\text{ m/s})^2}{2(9.81\text{ m/s}^2)}=1.52\text{ m}$$

$$E_{s2}=E_{s1}-\Delta z_b=1.52-0.22=1.30\text{ m}$$

$$E_c=\frac{3}{2}y_c=1.46\text{ m}$$

We have an interesting situation: The calculations show that $E_{s2} < E_c$. That is, the specific energy of the fluid decreases below the level of energy at the critical point, which is the minimum energy, and this is impossible. Therefore, the flow at specified conditions cannot exist. **The flow is choked** when the specific energy drops to the minimum value of 1.46 m, which occurs at a bump-height of $\Delta z_{b,\max}=E_{s1}-E_c=1.52-1.46=0.06\text{ m}$ .

***Discussion*** A bump-height over 6 cm results in a reduction in the flow rate of water, or a rise of upstream water level. Therefore, a 22-cm high bump alters the upstream flow. On the other hand, a bump less than 6 cm high will not affect the upstream flow.

---

**13-97**

**Solution** Water flowing in a horizontal open channel encounters a bump. The change in the surface level over the bump and the type of flow (sub- or supercritical) over the bump are to be determined.



***Assumptions*** **1** The flow is steady. **2** Frictional effects are negligible so that there is no dissipation of mechanical energy. **3** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The upstream Froude number and the critical depth are

$$\mathrm{Fr}_1 = \frac{V_1}{\sqrt{gy_1}} = \frac{8\ \mathrm{m/s}}{\sqrt{(9.81\ \mathrm{m^2/s})(0.8\ \mathrm{m})}} = 2.856$$

$$y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{(by_1V_1)^2}{gb^2}\right)^{1/3} = \left(\frac{y_1^2V_1^2}{g}\right)^{1/3} = \left(\frac{(0.8\ \mathrm{m})^2(8\ \mathrm{m/s})^2}{9.81\ \mathrm{m/s^2}}\right)^{1/3} = 1.61\ \mathrm{m}$$

The upstream flow is supercritical since Fr > 1, and the flow depth increases over the bump. The upstream, over the bump, and critical specific energies are

$$E_{s1} = y_1 + \frac{V_1^2}{2g} = (0.8\ \mathrm{m}) + \frac{(8\ \mathrm{m/s})^2}{2(9.81\ \mathrm{m/s^2})} = 4.06\ \mathrm{m}$$

$$E_{s2} = E_{s1} - \Delta z_b = 4.06 - 0.30 = 3.76\ \ \mathrm{m}$$

$$E_c = \frac{3}{2}y_c = 2.42\ \ \mathrm{m}$$

The flow depth over the bump is determined from

$$y_2^3 - (E_{s1} - \Delta z_b)y_2^2 + \frac{V_1^2}{2g}y_1^2 = 0 \quad \rightarrow \quad y_2^3 - (4.06 - 0.30\ \mathrm{m})y_2^2 + \frac{(8\ \mathrm{m/s})^2}{2(9.81\ \mathrm{m/s^2})}(0.80\ \mathrm{m})^2 = 0$$

Using an equation solver, the physically meaningful root of this equation is determined to be 0.846 m. Therefore, there is a rise of

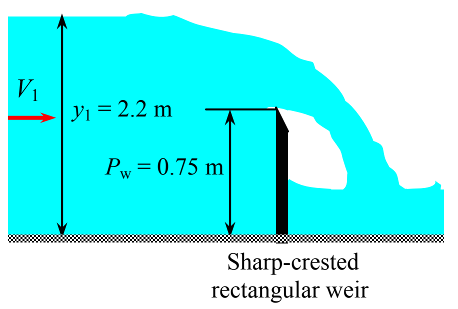
$$\text{Rise over bump} = y_2 - y_1 + \Delta z_b = 0.846 - 0.80 + 0.30 = \mathbf{0.346\ m}$$

over the surface relative to the upstream water surface. The specific energy decreases over the bump from, 4.06 to 3.76 m, but it is still over the minimum value of 2.42 m. Therefore, the flow over the bump is still **supercritical**.

***Discussion*** The actual value of surface rise may be different than 4.6 cm because of frictional effects that are neglected in this simplified analysis.

**13-98**

**Solution** The flow rate in an open channel is to be measured using a sharp-crested rectangular weir. For a measured value of flow depth upstream, the flow rate is to be determined.




***Assumptions*** **1** The flow is steady. **2** The upstream velocity head is negligible. **3** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The weir head is

$$H = y_1 - P_w = 2.2 - 0.75 = 1.45\text{ m}$$

The discharge coefficient of the weir is

$$C_{wd,\text{rec}} = 0.598 + 0.0897\frac{H}{P_w} = 0.598 + 0.0897\frac{1.45\text{ m}}{0.75\text{ m}} = 0.771$$

The condition $H/P_w < 2$ is satisfied since 1.45/0.75 = 1.93. Then the water flow rate through the channel becomes

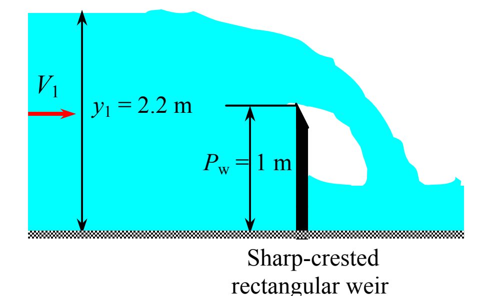
$$\dot{V}_{\text{rec}} = C_{wd,\text{rec}}\frac{2}{3}b\sqrt{2g}H^{3/2} = (0.7714)\frac{2}{3}(4\text{ m})\sqrt{2(9.81\text{ m/s}^2)}(1.45\text{ m})^{3/2} = \mathbf{15.9\ m^3/s}$$

***Discussion*** The upstream velocity and the upstream velocity head are $V_1 = \frac{\dot{V}}{by_1} = \frac{15.9\text{ m}^3/\text{s}}{(4\text{ m})(2.2\text{ m})} = 1.81\text{ m/s}$ and $\frac{V_1^2}{2g} = \frac{(1.81\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 0.167\text{ m}$ respectively. This is 11.5% of the weir head, which is significant. When the upstream velocity head is considered, the flow rate becomes 18.1 m$^3$/s, which is about 14 percent higher than the value determined above. Therefore, it is good practice to consider the upstream velocity head unless the weir height $P_w$ is very large relative to the weir head $H$.

---

**13-99**

**Solution** The flow rate in an open channel is to be measured using a sharp-crested rectangular weir. For a measured value of flow depth upstream, the flow rate is to be determined.




***Assumptions*** **1** The flow is steady. **2** The upstream velocity head is negligible. **3** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The weir head is $H = y_1 - P_w = 2.2 - 1.0 = 1.2\text{ m}$. The discharge coefficient of the weir is

$$C_{wd,\text{rec}} = 0.598 + 0.0897\frac{H}{P_w} = 0.598 + 0.0897\frac{1.2\text{ m}}{1.0\text{ m}} = 0.7056$$

The condition $H/P_w < 2$ is satisfied since 1.2/1.0 = 1.20. Then the water flow rate through the channel becomes

$$\dot{V}_{\text{rec}} = C_{wd,\text{rec}}\frac{2}{3}b\sqrt{2g}H^{3/2} = (0.7056)\frac{2}{3}(4\text{ m})\sqrt{2(9.81\text{ m/s}^2)}(1.2\text{ m})^{3/2} = \mathbf{11.0\ m^3/s}$$

***Discussion*** The upstream velocity and the upstream velocity head are $V_1 = \frac{\dot{V}}{by_1} = \frac{11.0\text{ m}^3/\text{s}}{(4\text{ m})(2.2\text{ m})} = 1.25\text{ m/s}$ and $\frac{V_1^2}{2g} = \frac{(1.25\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 0.079\text{ m}$, respectively. This is 6.6% of the weir head, which may be significant. When the upstream velocity head is considered, the flow rate becomes 11.9 m$^3$/s, which is about 8 percent higher than the value determined above. Therefore, it is good practice to consider the upstream velocity head unless the weir height $P_w$ is very large relative to the weir head $H$.

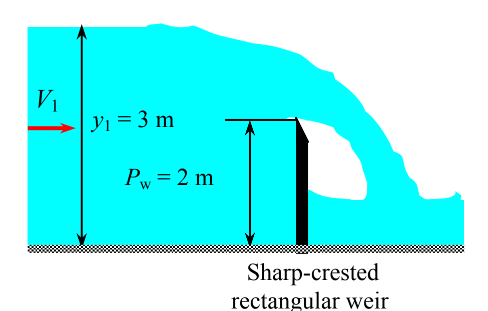
---

## 13-100

**Solution** Water flowing over a sharp-crested rectangular weir is discharged into a channel where uniform flow conditions are established. The maximum slope of the downstream channel to avoid hydraulic jump is to be determined.

***Assumptions*** **1** The flow is steady. **2** The upstream velocity head is negligible. **3** The channel is sufficiently wide so that the end effects are negligible.

***Properties*** Manning coefficient for an open channel of unfinished concrete is $n = 0.014$ (Table 13-1).



Sharp-crested rectangular weir

***Analysis*** The weir head is $H = y_1 - P_w = 3.0\text{ m} - 2.0\text{ m} = 1.0\text{ m}$. The condition $H/P_w < 2$ is satisfied since 1.0/2.0 = 0.5. The discharge coefficient of the weir is

$$C_{wd,\text{rec}} = 0.598 + 0.0897\frac{H}{P_w} = 0.598 + 0.0897\frac{1.0\text{ m}}{2.0\text{ m}} = 0.6429$$

Then the water flow rate through the channel per meter width (i.e., taking $b = 1$ m) becomes

$$\dot{V}_{\text{rec}} = C_{wd,\text{rec}}\frac{2}{3}b\sqrt{2g}H^{3/2} = (0.6429)\frac{2}{3}(1\text{ m})\sqrt{2(9.81\text{ m/s}^2)}(1.0\text{ m})^{3/2} = 1.898\text{ m}^3/\text{s}$$

To avoid hydraulic jump, we must avoid supercritical flow in the channel. Therefore, the bottom slope should not be higher than the critical slope, in which case the flow depth becomes the critical depth,

$$y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{(1.898\text{ m}^3/\text{s})^2}{(9.81\text{ m/s}^2)(1\text{ m}^2)}\right)^{1/3} = 0.7162\text{ m}$$

Noting that the hydraulic radius of a wide channel is equal to the flow depth, the bottom slope is determined from the Manning equation to be

$$\dot{V} = \frac{a}{n}A_c R_h^{2/3} S_0^{1/2} \quad\rightarrow\quad 1.898\text{ m}^3/\text{s} = \frac{1\text{ m}^{1/3}/s}{0.014}(0.7162\times 1\text{ m}^2)(0.7162\text{ m})^{2/3}S_0^{1/2}$$

Solution gives the slope to be $S_0 = 0.00215$. Therefore, $S_{0,\max} =$ **0.00215**.

***Discussion*** For a bottom slope smaller than calculated value, downstream channel would have a mild slope, that will force the flow to remain subcritical.

---

## 13-101E

**Solution** The flow rate in an open channel is to be measured using a sharp-crested rectangular weir. For specified upper limits of flow rate and flow depth, the appropriate height of the weir is to be determined.

***Assumptions*** **1** The flow is steady. **2** The upstream velocity head is negligible. **3** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The weir head is $H = y_1 - P_w = 5 - P_w$. The discharge coefficient of the weir is

$$C_{wd,\text{rec}} = 0.598 + 0.0897\frac{H}{P_w} = 0.598 + 0.0897\frac{5 - P_w}{P_w}$$

The water flow rate through the channel can be expressed as

$$\dot{V}_{\text{rec}} = C_{wd,\text{rec}}\frac{2}{3}b\sqrt{2g}H^{3/2}$$

Substituting the known quantities,

$$150\text{ ft}^3/\text{s} = \left(0.598 + 0.0897\frac{5 - P_w}{P_w}\right)\frac{2}{3}(10\text{ ft})\sqrt{2(32.2\text{ ft/s}^2)}(5 - P_w)^{3/2}$$

$V_1$ $y_1 = 5$ ft $P_w$

Sharp-crested rectangular weir

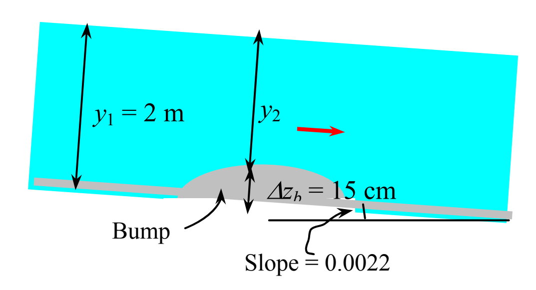
Solution of the above equation yields the weir height as $P_w =$ **2.46 ft**.

***Discussion*** Nonlinear equations of this kind can be solved easily using equation solvers like EES.

---

**13-102**

**Solution** The flow of water in a wide channel with a bump is considered. The flow rate of water without the bump and the effect of the bump on the flow rate for the case of a flat surface are to be determined.




***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel. **4** The channel is sufficiently wide so that the end effects are negligible. **5** Frictional effects during flow over the bump are negligible.

***Properties*** Manning coefficient for an open channel of unfinished concrete is $n$ = 0.014 (Table 13-1).

***Analysis*** For a wide channel, the hydraulic radius is equal to the flow depth, and thus $R_h$ = 2 m. Then the flow rate *before the bump* per m width (i.e., $b$ = 1 m) can be determined from Manning's equation to be

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} = \frac{1\,\text{m}^{1/3}/s}{0.014}(1\times 2\,\text{m}^2)(2\,\text{m})^{2/3}(0.0022)^{1/2} = 10.64\,\text{m}^3/\text{s}$$

The average flow velocity is $V = \dfrac{\dot{V}}{A_c} = \dfrac{10.64\,\text{m}^3/\text{s}}{1\times 2\,\text{m}^2} = 5.32\,\text{m/s}$ .

When a bump is placed, it is said that the flow depth remains the same and there is no rise/drop, and thus $y_2 = y_1 - \Delta z_b$ . But the energy equation is given as

$$E_{s2} = E_{s1} - \Delta z_b \quad \rightarrow \quad y_2 + \frac{V_2^2}{2g} = y_1 + \frac{V_1^2}{2g} - \Delta z_b \quad \rightarrow \quad \frac{V_2^2}{2g} = \frac{V_1^2}{2g}$$

since $y_2 = y_1 - \Delta z_b$ , and thus $V_1 = V_2$. But from the continuity equation $y_2 V_2 = y_1 V_1$ , this is possible only if the flow depth over the bump remains constant, i.e., $y_1 = y_2$, which is a contradiction since $y_2$ cannot be equal to both $y_1$ and $y_1 - \Delta z_b$ while $\Delta z_b$ remains nonzero. Therefore, the second part of the problem can have **no solution** since it is physically impossible.

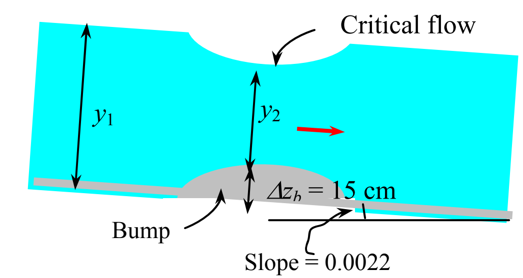
***Discussion*** Note that sometimes it is better to investigate whether there is really a solution before spending a lot of time trying to find a solution.

**13-103**

**Solution** Uniform subcritical water flow of water in a wide channel with a bump is considered. For critical flow over the bump, the flow rate of water and the flow depth over the bump are to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel. **4** The channel is sufficiently wide so that the end effects are negligible. **5** Frictional effects during flow over the bump are negligible.

***Properties*** Manning coefficient for an open channel of unfinished concrete is $n = 0.014$ (Table 13-1).




***Analysis*** Let subscript 1 denote the upstream conditions (uniform flow) in the channel, and 2 denote the critical conditions over the bump. For a wide channel, the hydraulic radius is equal to the flow depth, and thus $R_h = y_1$. Then the flow rate per m width (i.e., $b = 1$ m) can be determined from Manning's equation,

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} = \frac{1\,\text{m}^{1/3}/s}{0.014} y_1 (y_1)^{2/3} (0.0022)^{1/2} = 3.350 y_1^{5/3} \text{ m}^3/\text{s}$$

The critical depth corresponding to this flow rate is (note that $b = 1$ m),

$$y_2 = y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{(3.350 y_1^{5/3})^2}{g}\right)^{1/3} = \left(\frac{11.224 y_1^{10/3}}{9.81\,\text{m/s}^2}\right)^{1/3} = 1.046 y_1^{10/9}$$

The average flow velocity is $V_1 = \dot{V}/A_c = 3.350 y_1^{5/3}/y_1 = 3.350 y_1^{2/3}$ m/s . Also,

$$E_{s1} = y_1 + \frac{V_1^2}{2g} = y_1 + \frac{(3.350 y_1^{2/3})^2}{2(9.81\,\text{m/s}^2)} = y_1 + 0.5720 y_1^{4/3}$$

$$E_{s2} = E_c = \frac{3}{2} y_c = \frac{3}{2}(1.046 y_1^{10/9}) = 1.569 y_1^{10/9}$$

Substituting these two relations into $E_{s2} = E_{s1} - \Delta z_b$ where $\Delta z_b = 0.15$ m gives

$$1.569 y_1^{10/9} = y_1 + 0.5720 y_1^{4/3} - 0.15$$

Using an equation solver such as EES or an iterative approach, the flow depth upstream is determined to be

$$y_1 = 2.947 \text{ m}$$

Then the flow rate and the flow depth over the bump becomes

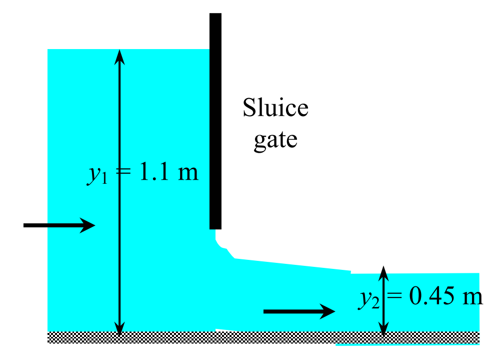
$$\dot{V} = 3.350 y_1^{5/3} = 3.350(2.947)^{5/3} = \mathbf{20.3\ m^3/s}$$

$$y_2 = y_c = 1.046 y_1^{10/9} = 1.046(2.947)^{10/9} = \mathbf{3.48\ m}$$

***Discussion*** Note that when critical flow is established and the flow is "choked", the flow rate calculations become very easy, and it required minimal measurements. Also, $V_1 = 3.350(2.947)^{2/3} = 6.89$ m/s and $\text{Fr}_1 = V_1/\sqrt{gy_1} = (6.89\,\text{m/s})/\sqrt{(9.81\,\text{m}^2/\text{s})(2.947\,\text{m})} = 1.28$ , and thus the upstream flow is supercritical.

**13-104**

**Solution** A sluice gate is used to control the flow rate of water in a channel. For specified flow depths upstream and downstream from the gate, the flow rate of water and the downstream Froude number are to be determined.




***Assumptions*** **1** The flow is steady. **2** The channel is sufficiently wide so that the end effects are negligible. **3** Frictional effects associated with sluice gate are negligible. **4** The channel is horizontal.

***Analysis*** When frictional effects are negligible and the flow section is horizontal, the specific energy remains constant, $E_{s1} = E_{s2}$. Then,

$$y_1 + \frac{V_1^2}{2g} = y_2 + \frac{V_2^2}{2g} \quad \rightarrow \quad 1.1\,\text{m} + \frac{\dot{V}^2}{2(9.81\,\text{m/s}^2)[(5\,\text{m})(1.1\,\text{m})]^2} = 0.45\,\text{m} + \frac{\dot{V}^2}{2(9.81\,\text{m/s}^2)[(5\,\text{m})(0.45\,\text{m})]^2}$$

Solving for the flow rate gives $\dot{V} = \mathbf{8.806\ m^3/s}$. The downstream velocity and Froude number are

$$V_2 = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{by_2} = \frac{8.806\,\text{m}^3/\text{s}}{(5\,\text{m})(0.45\,\text{m})} = 3.914\,\text{m/s} \quad \text{and} \quad \text{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{3.914\,\text{m/s}}{\sqrt{(9.81\,\text{m/s}^2)(0.45\,\text{m})}} = \mathbf{1.86}$$

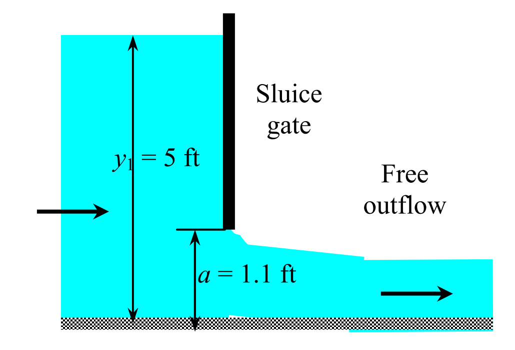
***Discussion*** The actual values will be somewhat different because of frictional effects.

---

**13-105E**

**Solution** Water is released from a reservoir through a sluice gate with free outflow. For specified flow depths, the flow rate per unit width and the downstream Froude number are to be determined.

***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible.




***Analysis*** For free outflow, we only need the depth ratio $y_1/a$ to determine the discharge coefficient (for drowned outflow, we also need to know $y_2/a$ and thus the flow depth $y_2$ downstream the gate)

$$\frac{y_1}{a} = \frac{5\,\text{ft}}{1.1\,\text{ft}} = 4.55$$

The corresponding discharge coefficient is determined from Fig. 13-38 to be $C_d$ = 0.55. Then the discharge rate becomes

$$\dot{V} = C_d ba\sqrt{2gy_1} = 0.55\,(1\,\text{ft})(1.1\,\text{ft})\sqrt{2\,(32.2\,\text{ft/s}^2)(5\,\text{ft})} = \mathbf{10.9\ \ ft^3/s}$$

The specific energy of a fluid remains constant during horizontal flow when the frictional effects are negligible, $E_{s1} = E_{s2}$. With these approximations, the flow depth past the gate and the Froude number are determined to be

$$E_{s1} = y_1 + \frac{V_1^2}{2g} = y_1 + \frac{\dot{V}^2}{2g(by_1)^2} = 5\,\text{ft} + \frac{(10.9\,\text{ft/s}^2)^2}{2(32.2\,\text{ft/s}^2)[(1\,\text{ft})(5\,\text{ft})]^2} = 5.074\,\text{ft}$$

$$E_{s2} = y_2 + \frac{V_2^2}{2g} = y_2 + \frac{\dot{V}^2}{2g(by_2)^2} = E_{s1} \quad \rightarrow \quad y_2 + \frac{(10.9\,\text{ft/s}^2)^2}{2(32.2\,\text{ft/s}^2)[(1\,\text{ft})(y_2)]^2} = 5.074\,\text{ft}$$

Solution yields $y_2$ = 0.643 ft as the physically meaningful root (positive and less than 5 ft). Then,

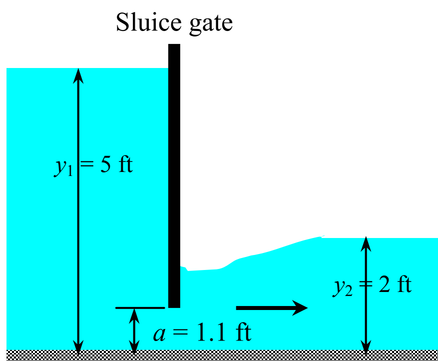
$$V_2 = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{by_2} = \frac{10.9\,\text{ft}^3/\text{s}}{(1\,\text{ft})(0.643\,\text{ft})} = 16.9\,\text{ft/s} \quad \text{and} \quad \text{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{16.9\,\text{ft/s}}{\sqrt{(32.2\,\text{ft/s}^2)(0.643\,\text{ft})}} = \mathbf{3.71}$$

***Discussion*** In actual gates some frictional losses are unavoidable, and thus the actual velocity and Froude number downstream will be lower.

---

**13-106E**

**Solution** Water is released from a reservoir through a drowned sluice gate into an open channel. For specified flow depths, the rate of discharge is to be determined.




***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The depth ratio $y_1/a$ and the contraction coefficient $y_2/a$ are

$$\frac{y_1}{a} = \frac{5 \text{ ft}}{1.1 \text{ ft}} = 4.55 \quad \text{and} \quad \frac{y_2}{a} = \frac{3.3 \text{ ft}}{1.1 \text{ ft}} = 3$$

The corresponding discharge coefficient is determined from Fig. 13-38 to be $C_d = 0.44$. Then the discharge rate becomes

$$\dot{V} = C_d ba\sqrt{2gy_1} = 0.44\,(1\text{ ft})(1.1\text{ ft})\sqrt{2\,(32.2\text{ ft/s}^2)(5\text{ ft})} = \mathbf{8.69\ ft^3/s}$$

Then the Froude number downstream the gate becomes

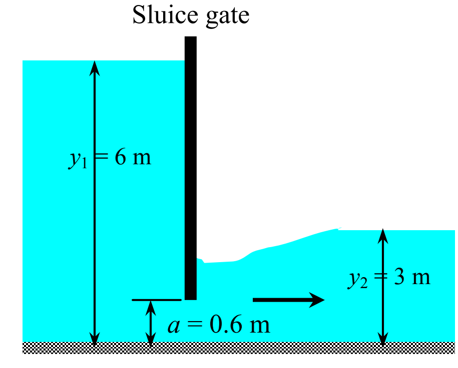
$$V_2 = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{by_2} = \frac{8.69\text{ ft}^3\text{/s}}{(1\text{ ft})(3.3\text{ ft})} = 2.63\text{ ft/s} \quad \rightarrow \quad \text{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{2.63\text{ ft/s}}{\sqrt{(32.2\text{ ft/s}^2)(3.3\text{ ft})}} = \mathbf{0.255}$$

***Discussion*** Note that the flow past the gate becomes subcritical when the outflow is drowned.

---

**13-107**

**Solution** Water is released from a lake through a drowned sluice gate into an open channel. For specified flow depths, the rate of discharge through the gate is to be determined.




***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The depth ratio $y_1/a$ and the contraction coefficient $y_2/a$ are

$$\frac{y_1}{a} = \frac{6 \text{ m}}{0.6 \text{ m}} = 10 \quad \text{and} \quad \frac{y_2}{a} = \frac{3 \text{ m}}{0.6 \text{ m}} = 5$$

The corresponding discharge coefficient is determined from Fig. 13-38 to be $C_d = 0.48$. Then the discharge rate becomes

$$\dot{V} = C_d ba\sqrt{2gy_1} = 0.48\,(5\text{ m})(0.6\text{ m})\sqrt{2\,(9.81\text{ m/s}^2)(6\text{ m})} = \mathbf{15.6\ m^3/s}$$

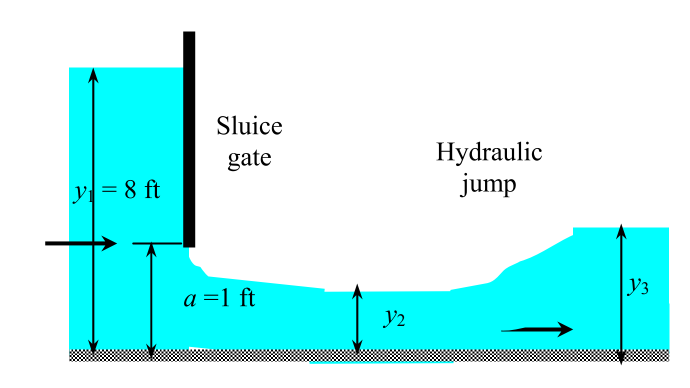
***Discussion*** Note that the use of the discharge coefficient enables us to determine the flow rate through sluice gates by measuring 3 flow depths only.

---

**13-108E**

**Solution** Water discharged through a sluice gate undergoes a hydraulic jump. The flow depth and velocities before and after the jump and the fraction of mechanical energy dissipated are to be determined.

***Assumptions*** **1** The flow is steady. **2** The channel is sufficiently wide so that the end effects are negligible. **3** Frictional effects associated with sluice gate are negligible. **4** The channel is horizontal.




***Analysis*** For free outflow, we only need the depth ratio $y_1/a$ to determine the discharge coefficient,

$$\frac{y_1}{a} = \frac{8\text{ ft}}{1\text{ ft}} = 8$$

The corresponding discharge coefficient is determined from Fig. 13-38 to be $C_d = 0.58$. Then the discharge rate becomes

$$\dot{V} = C_d ba\sqrt{2gy_1} = 0.58\,(1\text{ ft})(1\text{ ft})\sqrt{2\,(32.2\text{ ft/s}^2)(8\text{ ft})} = \mathbf{13.16\ ft^3/s}$$

The specific energy of a fluid remains constant during horizontal flow when the frictional effects are negligible, $E_{s1} = E_{s2}$. With these approximations, the flow depth past the gate and the Froude number are determined to be

$$E_{s1} = y_1 + \frac{V_1^2}{2g} = y_1 + \frac{\dot{V}^2}{2g(by_1)^2} = 8\text{ ft} + \frac{(13.16\text{ ft}^3/\text{s})^2}{2(32.2\text{ ft/s}^2)[(1\text{ ft})(8\text{ ft})]^2} = 8.042\text{ ft}$$

$$E_{s2} = y_2 + \frac{V_2^2}{2g} = y_2 + \frac{\dot{V}^2}{2g(by_2)^2} = E_{s1} \rightarrow y_2 + \frac{(13.16\text{ ft}^3/\text{s})^2}{2(32.2\text{ ft/s}^2)[(1\text{ ft})(y_2)]^2} = 8.042\text{ ft}$$

It gives $\mathbf{y_2 = 0.601}$ ft as the physically meaningful root (positive and less than 8 ft). Then,

$$V_2 = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{by_2} = \frac{13.16\text{ ft}^3/\text{s}}{(1\text{ ft})(0.601\text{ ft})} = \mathbf{21.9\ ft/s}$$

$$\text{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{21.9\text{ ft/s}}{\sqrt{(32.2\text{ ft/s}^2)(0.601\text{ ft})}} = 4.97$$

Then the flow depth and velocity after the jump (state 3) become

$$y_3 = 0.5y_2\left(-1+\sqrt{1+8\text{Fr}_2^2}\right) = 0.5(0.601\text{ ft})\left(-1+\sqrt{1+8\times 4.97^2}\right) = \mathbf{3.94\ ft}$$

$$V_3 = \frac{y_2}{y_3}V_2 = \frac{0.601\text{ ft}}{3.94\text{ ft}}(21.9\text{ ft/s}) = \mathbf{3.34\ ft/s}$$

The head loss and the fraction of mechanical energy dissipated during the jump are

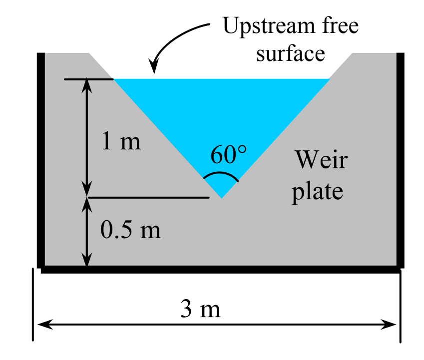
$$h_L = y_2 - y_3 + \frac{V_2^2 - V_3^2}{2g} = (0.601\text{ ft}) - (3.94\text{ ft}) + \frac{(21.9\text{ ft/s})^2 - (3.34\text{ ft/s})^2}{2(32.2\text{ ft/s}^2)} = 3.93\text{ ft}$$

$$\text{Dissipation ratio} = \frac{h_L}{E_{s2}} = \frac{h_L}{y_2(1+\text{Fr}_2^2/2)} = \frac{3.93\text{ ft}}{(0.601\text{ ft})(1+4.97^2/2)} = \mathbf{0.488}$$

***Discussion*** Note that almost half of the mechanical energy of the fluid is dissipated during hydraulic jump.

**13-109**

**Solution** The flow rate of water in an open channel is to be measured with a sharp-crested triangular weir. For a given flow depth upstream the weir, the flow rate is to be determined.




***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible.

***Properties*** The weir discharge coefficient is given to be 0.60.

***Analysis*** The discharge rate of water is determined directly from

$$\dot{V} = C_{wd,\text{tri}} \frac{8}{15} \tan\left(\frac{\theta}{2}\right) \sqrt{2g} H^{5/2}$$

where $C_{wd}$ = 0.60, $\theta$ = 60°, and $H$ = 1 m. Substituting,

$$\dot{V} = (0.60) \frac{8}{15} \tan\left(\frac{60°}{2}\right) \sqrt{2(9.81\,\text{m/s}^2)} (1\,\text{m})^{5/2} = \mathbf{0.818\ m^3/s}$$

***Discussion*** Note that the use of the discharge coefficient enables us to determine the flow rate in a channel by measuring a single flow depth. Triangular weirs are best-suited to measure low discharge rates as they are more accurate than the other weirs for small heads.

---

**13-110**

**Solution** The flow rate of water in an open channel is to be measured with a sharp-crested triangular weir. For a given flow depth upstream the weir, the flow rate is to be determined.

Upstream free surface
0.7 m
60°
Weir plate
0.5 m
3 m

***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible.

***Properties*** The weir discharge coefficient is given to be 0.60.

***Analysis*** The discharge rate of water is determined directly from

$$\dot{V} = C_{wd,\text{tri}} \frac{8}{15} \tan\left(\frac{\theta}{2}\right) \sqrt{2g} H^{5/2}$$

where $C_{wd}$ = 0.60, $\theta$ = 60°, and $H$ = 0.7 m. Substituting,

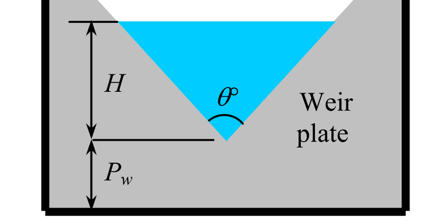
$$\dot{V} = (0.60) \frac{8}{15} \tan\left(\frac{60°}{2}\right) \sqrt{2(9.81\,\text{m/s}^2)} (0.7\,\text{m})^{5/2} = \mathbf{0.335\ m^3/s}$$

***Discussion*** Note that the use of the discharge coefficient enables us to determine the flow rate in a channel by measuring a single flow depth. Triangular weirs are best-suited to measure low discharge rates as they are more accurate than the other weirs for small heads.

---

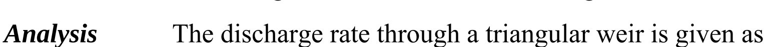
**13-111**

**Solution** The notch angle of a sharp-crested triangular weir used to measure the discharge rate of water from a lake is reduced by half. The percent reduction in the discharge rate is to be determined.




***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible. **3** The water depth in the lake and the weir discharge coefficient remain unchanged.

***Analysis*** The discharge rate through a triangular weir is given as



$$\dot{V} = C_{wd,\text{tri}} \frac{8}{15} \tan\left(\frac{\theta}{2}\right)\sqrt{2g}H^{5/2}$$

Therefore, the discharge rate is proportional to the tangent of the half notch angle, and the ratio of discharge rates is calculated to be

$$\dot{V} = \frac{\dot{V}_{50°}}{\dot{V}_{100°}} = \frac{\tan(50°/2)}{\tan(100°/2)} = 0.391$$

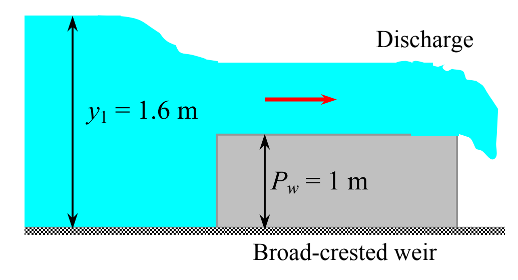
When the notch angle is reduced by half, the discharge rate drops to 39.1% of the original level. Therefore, the percent reduction in the discharge rate is

Percent reduction = 1-0.391 =0.609= **60.9%**

***Discussion*** Note that triangular weirs with small notch angles can be used to measure small discharge rates while weirs with large notch angles can be used to measure for large discharge rates.

**13-112**

**Solution** The flow rate in an open channel is to be measured using a broad-crested rectangular weir. For a measured value of flow depth upstream, the flow rate is to be determined.




***Assumptions*** **1** The flow is steady. **2** The upstream velocity head is negligible. **3** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The weir head is $H = y_1 - P_w = 1.6 - 1.0 = 0.6\text{ m}$. The discharge coefficient of the weir is

$$C_{wd,\text{broad}} = \frac{0.65}{\sqrt{1+H/P_w}} = \frac{0.65}{\sqrt{1+(0.6\text{ m})/(1.0\text{ m})}} = 0.5139$$

Then the water flow rate through the channel becomes

$$\dot{V}_{\text{rec}} = C_{wd,\text{broad}} b\sqrt{g}\left(\frac{2}{3}\right)^{3/2} H^{3/2} = (0.5139)(5\text{ m})(2/3)^{3/2}\sqrt{9.81\text{ m/s}^2}(0.6\text{ m})^{3/2} = \mathbf{2.04\ m^3/s}$$

The minimum flow depth above the weir is the critical depth, which is determined from

$$y_{\min} = y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{(2.04\text{ m}^3\text{/s})^2}{(9.81\text{ m/s}^2)(5\text{ m})^2}\right)^{1/3} = \mathbf{0.257\ m}$$

***Discussion*** The upstream velocity and the upstream velocity head are

$$V_1 = \frac{\dot{V}}{by_1} = \frac{2.04\text{ m}^3\text{/s}}{(5\text{ m})(1.6\text{ m})} = 0.255\text{ m/s} \quad \text{and} \quad \frac{V_1^2}{2g} = \frac{(0.255\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 0.0033\text{ m}$$

This is 0.3% of the weir head, which is negligible. When the upstream velocity head is considered (by replacing $H$ in the flow rate relation by $H + V_1^2/2g$), the flow rate becomes 2.05 m$^3$/s, which is practically identical to the value determined above.

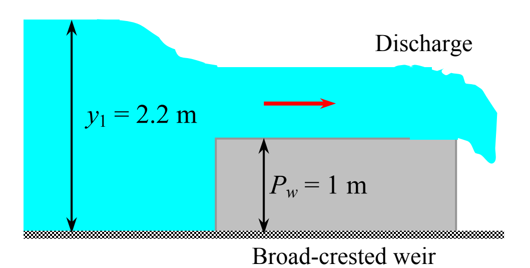
**13-113**

**Solution** The flow rate in an open channel is to be measured using a broad-crested rectangular weir. For a measured value of flow depth upstream, the flow rate is to be determined.

***Assumptions*** **1** The flow is steady. **2** The upstream velocity head is negligible. **3** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The weir head is $H = y_1 - P_w = 2.2 - 1.0 = 1.2\text{ m}$. The discharge coefficient of the weir is

$$C_{wd,\text{broad}} = \frac{0.65}{\sqrt{1+H/P_w}} = \frac{0.65}{\sqrt{1+(1.2\text{ m})/(1.0\text{ m})}} = 0.4382$$




Then the water flow rate through the channel becomes

$$\dot{V}_\text{rec} = C_{wd,\text{broad}} b\sqrt{g}\left(\frac{2}{3}\right)^{3/2} H^{3/2}$$

$$= (0.4382)(5\text{ m})(2/3)^{3/2}\sqrt{9.81\text{ m/s}^2}(1.2\text{ m})^{3/2}$$

$$= \mathbf{4.91\ m^3/s}$$

The minimum flow depth above the weir is the critical depth, which is determined from

$$y_\text{min} = y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{(4.91\text{ m}^3/\text{s})^2}{(9.81\text{ m/s}^2)(5\text{ m})^2}\right)^{1/3} = \mathbf{0.462\ m}$$

***Discussion*** The upstream velocity and the upstream velocity head are

$$V_1 = \frac{\dot{V}}{by_1} = \frac{4.91\text{ m}^3/\text{s}}{(5\text{ m})(2.2\text{ m})} = 0.446\text{ m/s} \quad \text{and} \quad \frac{V_1^2}{2g} = \frac{(0.446\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 0.010\text{ m}$$

This is 0.8% of the weir head, which is negligible. When the upstream velocity head is considered (by replacing $H$ in the flow rate relation by $H + V_1^2/2g$, the flow rate becomes 4.97 m$^3$/s, which is practically identical to the value determined above.

**13-114**

**Solution** The flow rate in an open channel is measured using a broad-crested rectangular weir. For a measured value of minimum flow depth over the weir, the flow rate and the upstream flow dept are to be determined.

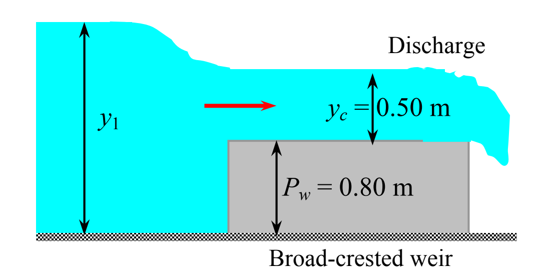
***Assumptions*** **1** The flow is steady. **2** The upstream velocity head is negligible. **3** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The flow depth over the reaches its minimum value when the flow becomes critical. Therefore, the measured minimum depth is the critical depth $y_c$. Then the flow rate is determined from the critical depth relation to be

$$y_{\min} = y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} \rightarrow \dot{V} = \sqrt{y_c^3 g b^2} = \sqrt{(0.50\text{ m})^3(9.81\text{ m/s}^2)(1\text{ m})^2} = \mathbf{1.11\ m^3/s}$$

This is the flow rate per m width of the channel since we have taken $b$ = 1 m. Disregarding the upstream velocity head and noting that the discharge coefficient of the weir is $C_{wd,\text{broad}} = 0.65/\sqrt{1+H/P_w}$, the flow rate for a broad-crested weir can be expressed as

$$\dot{V}_{\text{rec}} = \frac{0.65}{\sqrt{1+H/P_w}}\, b\sqrt{g}\left(\frac{2}{3}\right)^{3/2} H^{3/2}$$




Substituting,

$$1.11\text{ m}^3\text{/s} = \frac{0.65\text{ m}}{\sqrt{1+\text{H}/(0.8\text{ m})}}(1\text{ m})(2/3)^{3/2}\sqrt{9.81\text{ m/s}^2}\, H^{3/2}$$
$$= \mathbf{4.91 m^3/s}$$

Its solution is $H$ = 1.40 m. Then the flow depth upstream the weir becomes

$$y_1 = H + P_w = 1.40 + 0.80 = \mathbf{2.20\ m}$$

***Discussion*** The upstream velocity and the upstream velocity head are

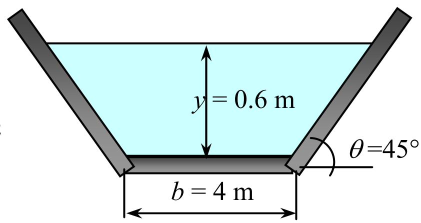
$$V_1 = \frac{\dot{V}}{by_1} = \frac{1.11\text{ m}^3\text{/s}}{(1\text{ m})(2.2\text{ m})} = 0.503\text{ m/s} \quad \text{and} \quad \frac{V_1^2}{2g} = \frac{(0.503\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 0.013\text{ m}$$

This is 0.9% of the weir head, which is negligible. When the upstream velocity head is considered (by replacing $H$ in the flow rate relation by $H + V_1^2/2g$, the flow rate becomes 1.12 m$^3$/s, which is practically identical to the value determined above.

## Review Problems

### 13-115

**Solution** Water flows uniformly in a trapezoidal channel. For a given flow depth, it is to be determined whether the flow is subcritical or supercritical.




***Assumptions*** The flow is uniform.

***Analysis*** The flow area and the average velocity are

$$A_c = y\frac{(b+b+2y/\tan\theta)}{2} = (0.60\text{ m})\frac{[4+4+2(0.60\text{ m})/\tan 45°]\text{ m}}{2} = 2.76\text{ m}^2$$

$$V = \frac{\dot{V}}{A_c} = \frac{18\text{ m}^3/\text{s}}{2.76\text{ m}^2} = 6.522\text{ m/s}$$

When calculating the Froude number, the hydraulic depth should be used rather than the maximum depth or the hydraulic radius. For a non-rectangular channel, hydraulic depth is defined as the ratio of the flow area to top width,

$$y = y_h = \frac{A_c}{\text{Top width}} = \frac{A_c}{b+2y/\tan\theta} = \frac{2.76\text{ m}^2}{(4+2\times 0.60/\tan 45°)\text{ m}} = 0.5308\text{ m}$$

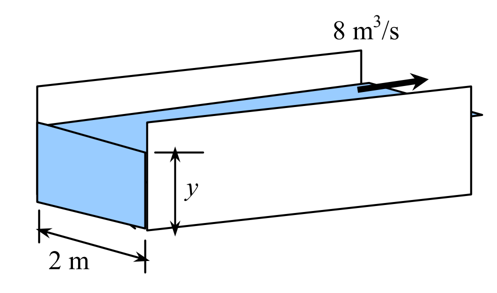
Then the Froude number becomes $\text{Fr} = \frac{V}{\sqrt{gy}} = \frac{6.522\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(0.5308\text{ m})}} = 2.86$, which is greater than 1.

Therefore, the flow is **supercritical**.

***Discussion*** The analysis is approximate since the edge effects are significant here compared to a wide rectangular channel, and thus the results should be interpreted accordingly.

### 13-116

**Solution** Water flows in a rectangular channel. The flow depth below which the flow is supercritical is to be determined.




***Assumptions*** The flow is uniform.

***Analysis*** The flow depth below which the flow is super critical is the critical depth $y_c$ determined from

$$y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{(8\text{ m}^3/\text{s})^2}{(9.81\text{ m/s}^2)(2\text{ m})^2}\right)^{1/3} = \mathbf{1.18\text{ m}}$$

Therefore, flow is **supercritical** for $y < 1.18$ m.

***Discussion*** Note that a flow is more likely to exist as supercritical when the flow depth is low and thus the flow velocity is high.

## 13-117

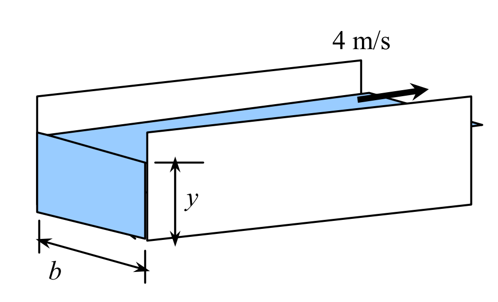
**Solution** Water flows in a canal at a specified average velocity. For various flow depths, it is to be determined whether the flow is subcritical or supercritical.

***Assumptions*** The flow is uniform.

***Analysis*** For each depth, we determine the Froude number and compare it to the critical value of 1:

(***a***) $y = 0.2$ m: $\quad \text{Fr} = \dfrac{V}{\sqrt{gy}} = \dfrac{4\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(0.2\text{ m})}} = 2.86 > 1$

which is greater than 1. Therefore, the flow is **supercritical**.




(***b***) $y = 2$ m: $\quad \text{Fr} = \dfrac{V}{\sqrt{gy}} = \dfrac{4\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(2\text{ m})}} = 0.903 < 1$

which is less than 1. Therefore, the flow is **subcritical**.

(***c***) $y = 1.63$ m: $\quad \text{Fr} = \dfrac{V}{\sqrt{gy}} = \dfrac{4\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(1.63\text{ m})}} = 1$

which is equal to 1. Therefore, the flow is **critical**.

***Discussion*** Note that a flow is more likely to exist as supercritical when the flow depth is low and thus the flow velocity is high. Also, the type of flow can be determined easily by checking Froude number.

---

## 13-118

**Solution** The flow of water in a rectangular channel is considered. For a given flow depth and bottom slope, the flow rate is to be determined.

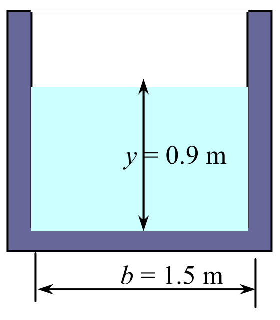
***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Properties*** The Manning coefficient is given to be $n = 0.012$ (Table 13-1).

***Analysis*** The flow area, wetted perimeter, and hydraulic radius of the channel are

$$A_c = by = (1.5\text{ m})(0.9\text{ m}) = 1.35\text{ m}^2 \qquad p = 1.5\text{ m} + 2\times 0.9\text{ m} = 3.3\text{ m}$$

$$R_h = \frac{A_c}{p} = \frac{1.35\text{ m}^2}{3.3\text{ m}} = 0.4091\text{ m}$$




Bottom slope of the channel is

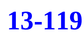
$$S_0 = \tan 0.6^\circ = 0.01047$$

Then the flow rate can be determined from Manning's equation to be

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} = \frac{1\text{ m}^{1/3}/s}{0.012}(1.35\text{ m}^2)(0.4091\text{ m})^{2/3}(0.01047)^{1/2} = \mathbf{6.34\ m^3/s}$$

***Discussion*** Note that the flow rate in a given channel is a strong function of the bottom slope.

---


**13-119** 

**Solution** The flow of water in a rectangular channel is considered. The effect of bottom slope on the flow rate is to be investigated as the bottom angle varies from 0.5 to 10°.

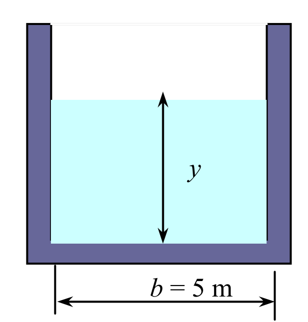
***Assumptions*** **1** The flow is steady and uniform. **2** Roughness coefficient is constant along the channel.

***Properties*** Manning coefficient for an open channel made of finished concrete is $n$ = 0.012 (Table 13-1).

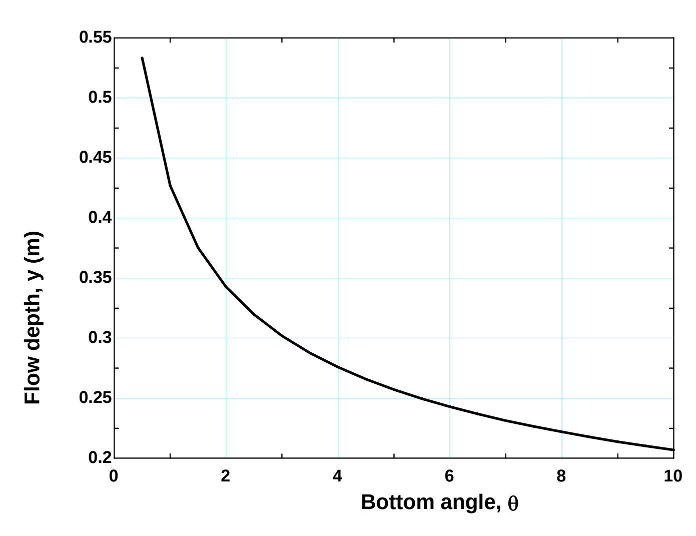
***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
a=1
b=5
Vdot = 12 "m3/s"
n=0.012

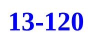
s=tan(teta)
Ac=b*y
p=b+2*y
Rh=Ac/p
Vdot=(a/n)*Ac*Rh^(2/3)*SQRT(s)
```




| Bottom angle, $\theta$° | Flow depth, $y$, m |
|---|---|
| 0.5 | 0.533 |
| 1.0 | 0.427 |
| 1.5 | 0.375 |
| 2.0 | 0.343 |
| 2.5 | 0.320 |
| 3.0 | 0.302 |
| 3.5 | 0.287 |
| 4.0 | 0.276 |
| 4.5 | 0.266 |
| 5.0 | 0.257 |
| 5.5 | 0.250 |
| 6.0 | 0.243 |
| 6.5 | 0.237 |
| 7.0 | 0.231 |
| 7.5 | 0.226 |
| 8.0 | 0.222 |
| 8.5 | 0.218 |
| 9.0 | 0.214 |
| 9.5 | 0.210 |
| 10.0 | 0.207 |




***Discussion*** Note that the flow depth decreases as the bottom angle increases, as expected.






**Solution** The flow of water in a trapezoidal channel is considered. The effect of bottom slope on the flow rate is to be investigated as the bottom angle varies from 0.5 to 10°.

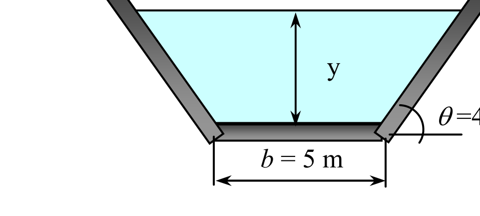
***Assumptions*** **1** The flow is steady and uniform. **2** Roughness coefficient is constant along the channel.

***Properties*** Manning coefficient for an open channel made of finished concrete is $n$ = 0.012 (Table 13-1).

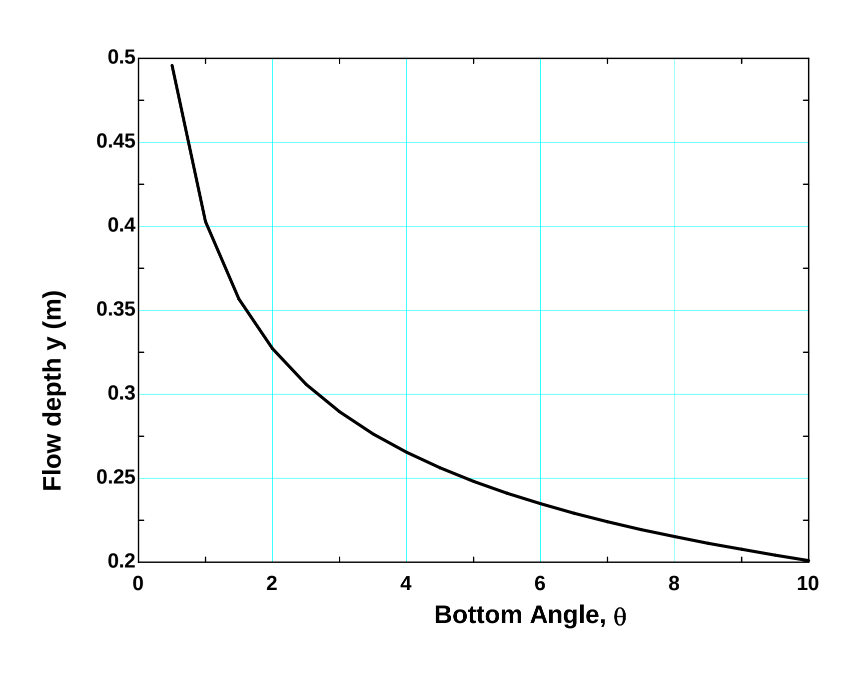
***Analysis*** The EES *Equations* window is printed below, along with the tabulated and plotted results.

```
a=1
b=5
Vdot = 12 "m3/s"
n=0.012

s=tan(teta)
Ac=y*(b+y/tan(45))
p=b+2*y/sin(45)
Rh=Ac/p
Vdot=(a/n)*Ac*Rh^(2/3)*SQRT(s)
```




| Bottom angle, $\theta°$ | Flow depth, $y$, m |
|---|---|
| 0.5 | 0.496 |
| 1.0 | 0.403 |
| 1.5 | 0.357 |
| 2.0 | 0.327 |
| 2.5 | 0.306 |
| 3.0 | 0.290 |
| 3.5 | 0.276 |
| 4.0 | 0.266 |
| 4.5 | 0.256 |
| 5.0 | 0.248 |
| 5.5 | 0.241 |
| 6.0 | 0.235 |
| 6.5 | 0.229 |
| 7.0 | 0.224 |
| 7.5 | 0.220 |
| 8.0 | 0.215 |
| 8.5 | 0.211 |
| 9.0 | 0.208 |
| 9.5 | 0.204 |
| 10.0 | 0.201 |




***Discussion*** As expected, flow depth decreases with increasing bottom angle, but the relationship is far from linear.

## 13-121

**Solution** The flow of water in a trapezoidal channel is considered. For a given flow depth and bottom slope, the flow rate is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Properties*** The Manning coefficient for a brick-lined open channel is $n$ = 0.015 (Table 13-1).

***Analysis*** The flow area, wetted perimeter, and hydraulic radius of the channel are

$$A_c = y\left(b+\frac{y}{\tan\theta}\right) = (2\text{ m})\left(4\text{ m}+\frac{2\text{ m}}{\tan 30°}\right) = 14.93\ \text{m}^2$$

$$p = b+\frac{2y}{\sin\theta} = 4\text{ m}+\frac{2(2\text{ m})}{\sin 30°} = 12\text{ m}$$

$$R_h = \frac{A_c}{p} = \frac{14.93\text{ m}^2}{12\text{ m}} = 1.244\ \text{m}$$

$y = 2$ m, 30°, $\theta = 30°$, $b = 4$

Bottom slope of the channel is $S_o$ = 0.001.Then the flow rate can be determined from Manning's equation to be

$$\dot{V} = \frac{a}{n}A_c R_h^{2/3} S_0^{1/2} = \frac{1\text{ m}^{1/3}/s}{0.015}(14.93\text{ m}^2)(1.244\text{ m})^{2/3}(0.001)^{1/2} = \mathbf{36.4\ m^3/s}$$

***Discussion*** Note that the flow rate in a given channel is a strong function of the bottom slope.

---

## 13-122

**Solution** The flow of water in a circular open channel is considered. For given flow depth and flow rate, the elevation drop per km length is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Properties*** The Manning coefficient for the steel channel is given to be $n$ = 0.012.

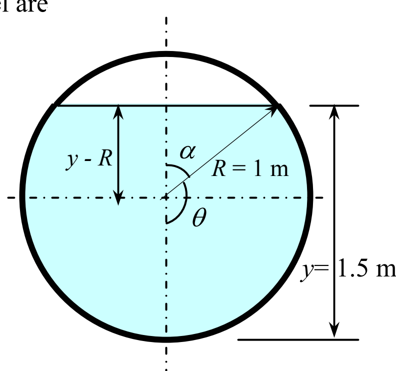
***Analysis*** The flow area, wetted perimeter, and hydraulic radius of the channel are

$$\cos\alpha = \frac{y-R}{R} = \frac{1.5-1}{1} = 0.5 \quad\rightarrow\quad \alpha = 60° = 60\frac{2\pi}{360} = \frac{\pi}{3}$$

$$\theta = \pi-\alpha = \pi-\frac{\pi}{3} = \frac{2\pi}{3} = 120°$$

$$A_c = R^2(\theta-\sin\theta\cos\theta) = (1\text{ m})^2[2\pi/3-\sin(2\pi/3)\cos(2\pi/3)] = 2.527\text{ m}^2$$

$$R_h = \frac{A_c}{p} = \frac{\theta-\sin\theta\cos\theta}{2\theta}R = \frac{2\pi/3-\sin(2\pi/3)\cos(2\pi/3)}{2\times 2\pi/3}(1\text{ m}) = 0.6034\text{ m}$$




Substituting the given quantities into Manning's equation,

$$\dot{V} = \frac{a}{n}A_c R_h^{2/3} S_0^{1/2} \quad\rightarrow\quad 12\text{ m}^3/\text{s} = \frac{1\text{ m}^{1/3}/s}{0.012}(2.527\text{ m}^2)(0.6034\text{ m})^{2/3} S_0^{1/2}$$

It gives the slope to be $S_0$ = 0.00637. Therefore, the *elevation drop* Δ*z* across a pipe length of *L* = 1 km must be

$$\Delta z = S_0 L = 0.00637(1000\text{ m}) = \mathbf{6.37\ m}$$

***Discussion*** Note that when transporting water through a region of fixed elevation drop, the only way to increase the flow rate is to use a channel with a larger cross-section.

---

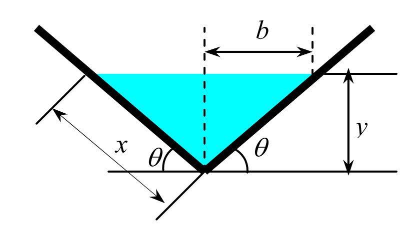
**13-123**

**Solution** The flow of water through a V-shaped open channel is considered. The angle θ the channel makes from the horizontal is to be determined for the case of most efficient flow.

***Assumptions*** **1** The flow is steady and uniform. **2** The bottom slope is constant. **3** The roughness coefficient is constant.

***Analysis*** We let the length of the sidewall of the channel be *x*. From trigonometry,

$$\sin\theta = \frac{y}{x} \quad \rightarrow \quad y = x\sin\theta \qquad \cos\theta = \frac{b}{x} \quad \rightarrow \quad b = x\cos\theta$$




Then the cross-sectional area and the perimeter of the flow section become

$$A_c = by = x\cos\theta\sin\theta = \frac{x^2}{2}\sin 2\theta \quad \rightarrow \quad x = \sqrt{\frac{2A_c}{\sin 2\theta}}$$

$$p = 2x = 2\sqrt{\frac{2A_c}{\sin 2\theta}} \quad \rightarrow \quad p = 2\sqrt{2A_c}(\sin 2\theta)^{-1/2}$$

Now we apply the criterion that the best hydraulic cross-section for an open channel is the one with the minimum wetted perimeter for a given cross-section. Taking the derivative of *p* with respect to *θ* while holding $A_c$ constant gives

$$\frac{dp}{d\theta} = 2\sqrt{2A_c}\,\frac{d[(\sin 2\theta)^{-1/2}]}{d\theta} = 2\sqrt{2A_c}\,\frac{d[(\sin 2\theta)^{-1/2}]}{d(\sin 2\theta)}\frac{d(\sin 2\theta)}{d\theta} = 2\sqrt{2A_c}\,\frac{-3}{2(\sin 2\theta)^{3/2}}2\cos 2\theta$$

Setting $dp/d\theta = 0$ gives $\cos 2\theta = 0$, which is satisfied when $2\theta = 90°$. Therefore, the criterion for the best hydraulic cross-section for a triangular channel is determined to be $\boldsymbol{\theta = 45°}$.

***Discussion*** The procedure used here can be used to determine the best hydraulic cross-section for any geometric shape.

**13-124E**

**Solution** Water is to be transported in a rectangular channel at a specified rate. The dimensions for the best cross-section if the channel is made of unfinished concrete are to be determined.

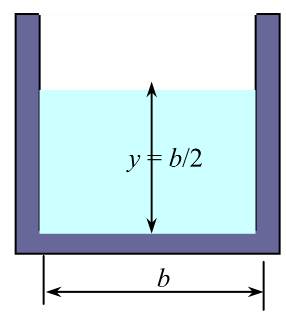
***Assumptions*** **1** The flow is steady and uniform. **2** The bottom slope is constant. **3** The roughness coefficient is constant.

***Properties*** The Manning coefficient is $n = 0.014$ for channels made of unfinished concrete (Table 13-1).

***Analysis*** For best cross-section of a rectangular cross-section, $y = b/2$. Then $A_c = yb = b^2/2$, and $R_h = b/4$.

The flow rate is determined from the Manning equation, $\dot{V} = \frac{a}{n}A_c R_h^{2/3} S_0^{1/2}$.

(*a*) Bottom drop of 8 ft per mile: $s = (8\text{ ft})/(5280\text{ ft}) = 0.001515$




$$200\text{ ft}^3/\text{s} = \frac{1.486\text{ ft}^{1/3}/s}{0.014}(b^2/2)(b/4)^{2/3}(0.001515)^{1/2}$$

Solving the above equation gives $b = \mathbf{7.86\text{ ft}}$, and $y = b/2 = \mathbf{3.93\text{ ft}}$.

(*b*) Bottom drop of 10 ft per mile: $s = (10\text{ ft})/(5280\text{ ft}) = 0.001894$

$$200\text{ ft}^3/\text{s} = \frac{1.486\text{ ft}^{1/3}/s}{0.014}(b^2/2)(b/4)^{2/3}(0.001894)^{1/2}$$

Solving the above equation gives $b = \mathbf{7.54\text{ ft}}$, and $y = b/2 = \mathbf{3.77\text{ ft}}$.

***Discussion*** The concept of best cross-section is an important consideration in the design of open channels because it directly affects the construction costs.



**13-125E**

**Solution** Water is to be transported in a trapezoidal channel at a specified rate. The dimensions for the best cross-section if the channel is made of unfinished concrete are to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Properties*** The Manning coefficient is $n$ = 0.014 for channels made of unfinished concrete (Table 13-1).

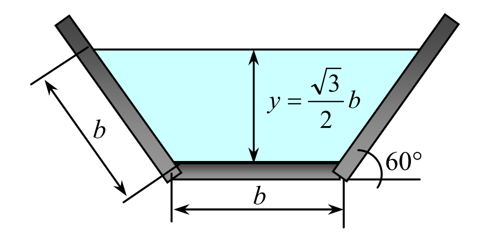
***Analysis*** For best cross-section of a trapezoidal channel of bottom width $b$, $\theta$ = 60° and $y = b\sqrt{3}/2$ . Then,

$$A_c = y(b + b\cos\theta) = 0.5\sqrt{3}b^2(1+\cos 60°) = 0.75\sqrt{3}b^2\,,\ \ p = 3b\,,\text{ and }\ R_h = \frac{y}{2} = \frac{\sqrt{3}}{4}b\,.$$

The flow rate is determined from the Manning equation, $\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2}$,

(*a*) Bottom drop of 8 ft per mile:

$$s = (8\text{ ft})/(5280\text{ ft}) = 0.001515$$




$$200\text{ ft}^3/\text{s} = \frac{1.486\text{ ft}^{1/3}/s}{0.014}(0.75\sqrt{3}b^2)(\sqrt{3}b/4)^{2/3}(0.001515)^{1/2}$$

It gives $b$ = **4.79 ft,** and $y$ = **4.15 ft.**

(*b*) Bottom drop of 10 ft per mile:

$$s = (10\text{ ft})/(5280\text{ ft}) = 0.001894$$

$$200\text{ ft}^3/\text{s} = \frac{1.486\text{ ft}^{1/3}/s}{0.014}(0.75\sqrt{3}b^2)(\sqrt{3}b/4)^{2/3}(0.001894)^{1/2}$$

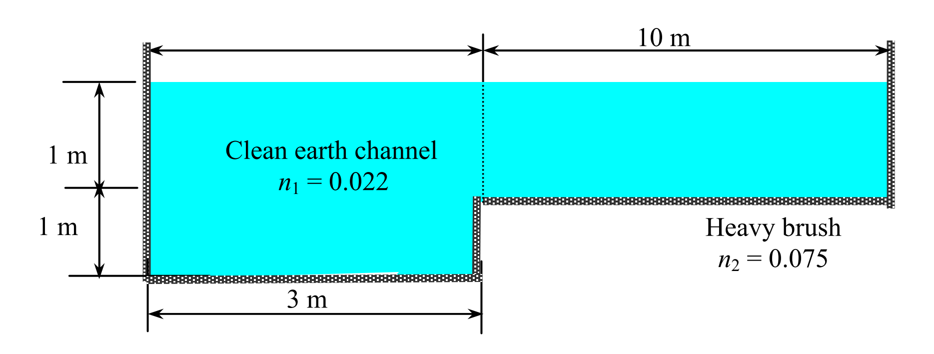
It gives $b$ = **4.59 ft,** and $y$ = **3.98 ft.**

***Discussion*** The concept of best cross-section is an important consideration in the design of open channels because it directly affects the construction costs.

**13-126**

**Solution** Water is flowing through a channel with nonuniform surface properties. The flow rate through the channel and the effective Manning coefficient are to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** The bottom slope is constant. **3** The Manning coefficients do not vary along the channel.




***Analysis*** The channel involves two parts with different roughness, and thus it is appropriate to divide the channel into two subsections. The flow rate for each subsection can be determined from the Manning equation, and the total flow rate can be determined by adding them up.

The flow area, perimeter, and hydraulic radius for each subsection and the entire channel are:

Subsection 1: $A_{c1} = 6\text{ m}^2$, $p_1 = 6\text{ m}$, $R_{h1} = \dfrac{A_{c1}}{p_1} = \dfrac{6\text{ m}^2}{6\text{ m}} = 1.00\text{ m}$

Subsection 2: $A_{c2} = 10\text{ m}^2$, $p_2 = 11\text{ m}$, $R_{h2} = \dfrac{A_{c2}}{p_2} = \dfrac{10\text{ m}^2}{11\text{ m}} = 0.909\text{ m}$

Entire channel: $A_c = 16\text{ m}^2$, $p = 17\text{ m}$, $R_h = \dfrac{A_c}{p} = \dfrac{16\text{ m}^2}{17\text{ m}} = 0.941\text{ m}$

Applying the Manning equation to each subsection, the total flow rate through the channel becomes

$$\dot{V} = \dot{V}_1 + \dot{V}_2 = \frac{a}{n_1} A_1 R_1^{2/3} S_0^{1/2} + \frac{a}{n_2} A_2 R_2^{2/3} S_0^{1/2}$$

$$= (1\text{ m}^{1/3}/\text{s})\left(\frac{(6\text{ m}^2)\ (1\text{ m})^{2/3}}{0.022} + \frac{(10\text{ m}^2)(0.909\text{ m})^{2/3}}{0.075}\right)(\tan 0.5°)^{1/2}$$

$$= \mathbf{37.2\ m^3/s}$$

Knowing the total flow rate, the effective Manning coefficient for the entire channel can be determined from the Manning equation to be

$$n_{\text{eff}} = \frac{aA_c R_h^{2/3} S_0^{1/2}}{\dot{V}} = \frac{(1\text{ m}^{1/3}/\text{s})(16\text{ m}^2)(0.941\text{ m})^{2/3}(0.00873)^{1/2}}{37.2\text{ m}^3/\text{s}} = \mathbf{0.0386}$$

***Discussion*** The effective Manning coefficient $n_{\text{eff}}$ of the channel turns out to lie between the two $n$ values, as expected. The weighted average of the Manning coefficient of the channel is $n_{\text{ave}}=(n_1p_1+ n_2p_2)/p$ = 0.056, which is quite different than $n_{\text{eff}}$. Therefore, using a weighted average Manning coefficient for the entire channel may be tempting, but it would not be so accurate.

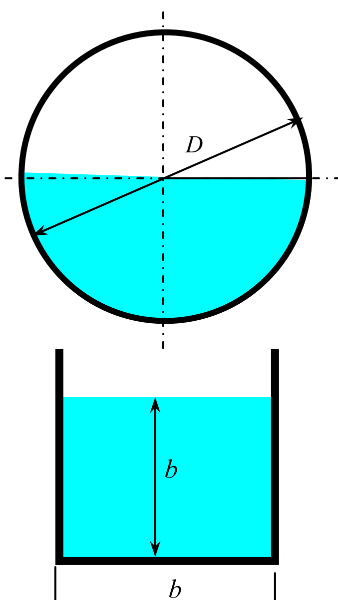
**13-127**

**Solution** Two identical channels, one rectangular of bottom width $b$ and one circular of diameter $D$, with identical flow rates, bottom slopes, and surface linings are considered. The relation between $b$ and $D$ is to be determined for the case of the flow height $y = b$ and the circular channel is flowing half full.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Analysis*** The cross-sectional area, perimeter, and hydraulic radius of the rectangular channel are

$$A_c = b^2, \quad p = 3b, \text{ and } \quad R_h = \frac{A_c}{p} = \frac{b^2}{3\text{b}} = \frac{b}{3}$$




Then using the Manning equation, the flow rate can be expressed as

$$\dot{V}_{\text{rec}} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} = \frac{a}{n} b^2 \left(\frac{b}{3}\right)^{2/3} S_0^{1/2} = \frac{a}{n} S_0^{1/2} \frac{b^{8/3}}{3^{2/3}}$$

The corresponding relations for the semi-circular channel are

$$A_c = \frac{\pi D^2}{8}, \quad p = \frac{\pi D}{2}, \text{ and } \quad R_h = \frac{A_c}{p} = \frac{D}{4}$$

and

$$\dot{V}_{\text{cir}} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} = \frac{a}{n} \pi \frac{D^2}{8} \left(\frac{D}{4}\right)^{2/3} S_0^{1/2} = \frac{a}{n} S_0^{1/2} \frac{\pi D^{8/3}}{8 \times 4^{2/3}}$$

Setting the flow rates in the two channels equal to each other $\dot{V}_{\text{cir}} = \dot{V}_{\text{rec}}$ gives

$$\frac{a}{n} S_0^{1/2} \frac{b^{8/3}}{3^{2/3}} = \frac{a}{n} \frac{\pi D^{8/3}}{8 \times 4^{2/3}} S_0^{1/2} \rightarrow \frac{b^{8/3}}{3^{2/3}} = \frac{\pi D^{8/3}}{8 \times 4^{2/3}} \rightarrow \frac{b}{D} = \left(\frac{\pi 3^{2/3}}{8 \times 4^{2/3}}\right)^{3/8} = 0.655$$

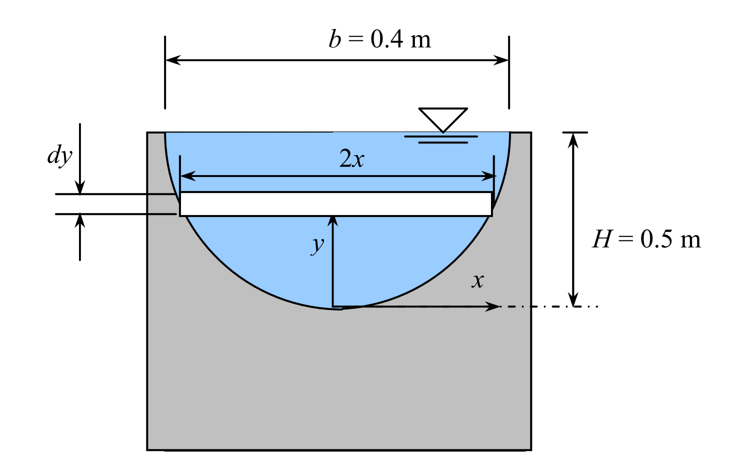
Therefore, the desired relation is **$b = 0.655D$**.

***Discussion*** Note that the wetted perimeters in this case are $p_{\text{rec}} = 3b = 2.0D$ and $p_{\text{cir}} = \pi D/2 = 1.57D$. Therefore, the semi-circular channel is a more efficient channel than the rectangular one.

**13-128**

**Solution** The flow of water through a parabolic notch is considered. A relation is to be developed for the flow rate, and its numerical value is to be calculated.

***Assumptions*** **1** The flow is steady. **2** All frictional effects are negligible, and Toricelli's equation can be used for the velocity.




***Analysis*** The notch is parabolic with $y = 0$ at $x = 0$, and thus it can be expressed analytically as $y = Cx^2$. Using the coordinates of the upper right corner, the value of the constant is determined to be

$$C = y/x^2 = H/(b/2)^2 = 4H/b^2 = 4(0.5\text{ m})/(0.4\text{ m})^2 = 12.5\text{ m}^{-1}.$$

A differential area strip can be expressed as

$$dA = 2xdy = 2\sqrt{y/C}\,dy$$

Noting that the flow velocity is $V = \sqrt{2g(H-y)}$, the flow rate through this differential area is

$$VdA = V\left(2\sqrt{y/C}\,dy\right) = \sqrt{2g(H-y)}\,2\sqrt{y/C}\,dy = 2\sqrt{2g/C}\sqrt{y(H-y)}\,dy$$

Then the flow rate through the entire notch is determined by integration to be

$$\dot{V} = \int_A VdA = 2\sqrt{2g/C}\int_{y=0}^{H}\sqrt{y(H-y)}\,dy$$

where

$$\int_{y=0}^{H}\sqrt{y(H-y)}\,dy = \left[\frac{1}{4}(2y-H)\sqrt{Hy-y^2} + \frac{H^2}{8}\,Arc\tan\left(\frac{2y-H}{2\sqrt{Hy-y^2}}\right)\right]_0^H = \frac{\pi}{16}H^2$$

Then the expression for the volume flow rate and its numerical value become

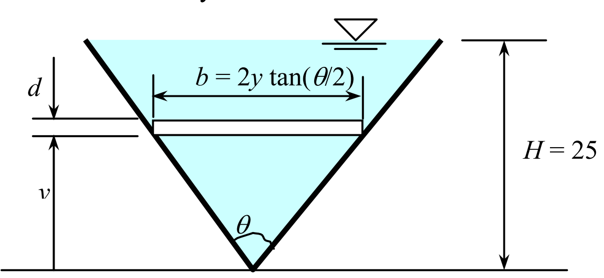
$$\boxed{\dot{V} = \frac{\pi}{8}\sqrt{\frac{2g}{C}}H^2 = \frac{\pi}{8}\sqrt{\frac{2(9.81\text{ m/s}^2)}{12.5\text{ m}^{-1}}}H^2 = (0.492\text{ m/s})H^2 = (0.492\text{ m/s})(0.5\text{ m})^2 = 0.123\text{ m}^3\text{/s}}$$

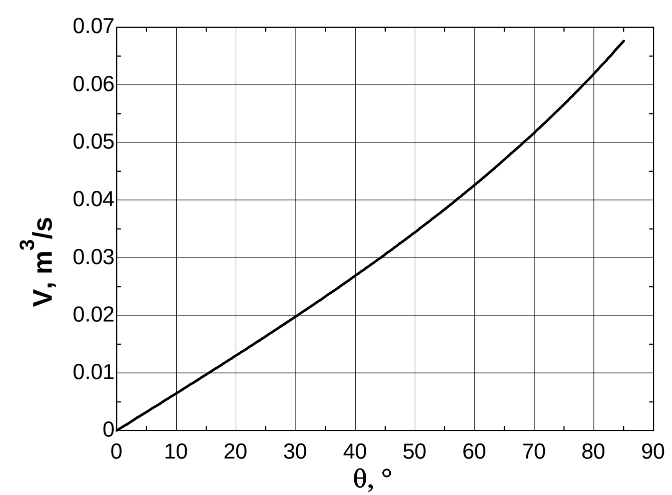
***Discussion*** Note that a general flow rate equation for parabolic notch would be in the form of $\dot{V} = KH^2$, where $K = C_d\dfrac{\pi}{8}\sqrt{\dfrac{2g}{C}}$ and $C_d$ is the discharge coefficient whose value is determined experimentally to account for nonideal effects.

**13-129** 

**Solution** The flow of water through a parabolic notch is considered. A relation is to be developed for the flow rate, and its numerical value is to be calculated.

***Assumptions*** **1** The flow is steady. **2** All frictional effects are negligible, and Toricelli's equation can be used for the velocity.




***Analysis*** Consider a differential strip area shown on the sketch. It can be expressed as

$$dA = bdy = 2y\tan(\theta/2)dy$$

Noting that the flow velocity is $V = \sqrt{2g(H-y)}$, the flow rate through this differential area is

$$VdA = V\big(2y\tan(\theta/2)dy\big) = \sqrt{2g(H-y)}\,2y\tan(\theta/2)dy = 2\sqrt{2g}\,\tan(\theta/2)y\sqrt{H-y}dy$$

Then the flow rate through the entire notch is determined by integration to be

$$\dot{V} = \int_A VdA = 2\sqrt{2g}\,\tan(\theta/2)\int_{y=0}^{H} y\sqrt{H-y}dy$$

where

$$\int_{y=0}^{H} y\sqrt{H-y}dy = \left[-\frac{2}{5}y^{5/2} + \frac{2}{3}Hy^{3/2}\right]\Bigg|_0^H = \frac{4}{15}H^{5/2}$$

Then the expression for the volume flow rate and its numerical value become

$$\dot{V} = \frac{8\sqrt{2g}}{15}\tan(\theta/2)H^{5/2} = \frac{8\sqrt{2(9.81\text{ m/s}^2)}}{15}\tan(\theta/2)(0.25)^{5/2} = 0.07382\tan(\theta/2) \quad (\text{m}^3/\text{s})$$

$\theta = 25°$: $\dot{V} = 0.07382\tan\left(25°/2\right) = \mathbf{0.0164\ m^3/s}$

$\theta = 40°$: $\dot{V} = 0.07382\tan\left(40°/2\right) = \mathbf{0.0269\ m^3/s}$

$\theta = 60°$: $\dot{V} = 0.07382\tan\left(60°/2\right) = \mathbf{0.0426\ m^3/s}$

$\theta = 75°$: $\dot{V} = 0.07382\tan\left(75°/2\right) = \mathbf{0.0566\ m^3/s}$

These results are plotted, using EES.

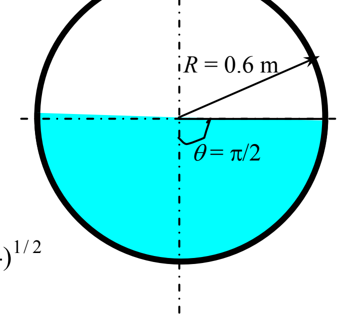
***Discussion*** Note that a general flow rate equation for the V-notch would be in the form of $\dot{V} = K\tan(\theta/2)H^{5/2}$, where $K = C_d 8\sqrt{2g}/15$ and $C_d$ is the discharge coefficient whose value is determined experimentally to account for nonideal effects.

**13-130**

**Solution** Water flows uniformly half-full in a circular channel. For specified flow rate and bottom slope, the Manning coefficient is to be determined.

***Assumptions*** **1** The flow is steady and uniform. **2** Bottom slope is constant. **3** Roughness coefficient is constant along the channel.

***Analysis*** The flow area, wetted perimeter, and hydraulic radius of the channel are




$$A_c = \frac{\pi R^2}{2} = \frac{\pi(0.6\text{ m})^2}{2} = 0.5655\text{ m}^2$$

$$p = \frac{2\pi R}{2} = \frac{2\pi(0.6\text{ m})}{2} = 1.885\text{ m}$$

$$R_h = \frac{A_c}{P} = \frac{\pi R^2/2}{\pi R} = \frac{R}{2} = \frac{0.60\text{ m}}{2} = 0.30\text{ m}$$

Then the Manning coefficient can be determined from Manning's equation to be

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} \quad \rightarrow \quad 1.25\text{ m}^3/\text{s} = \frac{1\text{ m}^{1/3}/s}{n}(0.5655\text{ m}^2)(0.30\text{ m})^{2/3}(0.004)^{1/2}$$

It gives the Manning coefficient to be

$$n = \mathbf{0.013}$$

When calculating the Froude number, the hydraulic depth should be used rather than the maximum depth or the hydraulic radius. For a non-rectangular channel, hydraulic depth is defined as the ratio of the flow area to top width,

$$y_h = \frac{A_c}{\text{Top width}} = \frac{\pi R^2/2}{2R} = \frac{\pi R}{4} = \frac{\pi(0.6\text{ m})}{4} = 0.4712\text{ m}$$

$$V = \frac{\dot{V}}{A_c} = \frac{1.25\text{ m}^3/\text{s}}{0.5655\text{ m}^2} = 2.210\text{ m/s}$$

$$\text{Fr} = \frac{V}{\sqrt{gy}} = \frac{2.21\text{m/s}}{\sqrt{(9.81\text{ m/s}^2)(0.4721\text{ m})}} = \mathbf{1.03}$$

which is greater than 1. Therefore, the flow is *supercritical*.

***Discussion*** It appears that this channel is made of cast iron or unplaned wood .

**13-131**

**Solution** Water flowing in a horizontal open channel encounters a bump. Flow properties over the bump are to be determined.

***Assumptions*** **1** The flow is steady. **2** Frictional effects are negligible so that there is no dissipation of mechanical energy. **3** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The upstream Froude number and the critical depth are

$$\mathrm{Fr}_1 = \frac{V_1}{\sqrt{gy_1}} = \frac{1.25\ \mathrm{m/s}}{\sqrt{(9.81\ \mathrm{m^2/s})(1.8\ \mathrm{m})}} = 0.297$$

$$y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{(by_1V_1)^2}{gb^2}\right)^{1/3} = \left(\frac{y_1^2V_1^2}{g}\right)^{1/3} = \left(\frac{(1.8\ \mathrm{m})^2(1.25\ \mathrm{m/s})^2}{9.81\ \mathrm{m/s^2}}\right)^{1/3} = 0.802\ \mathrm{m}$$

The upstream flow is subcritical since Fr < 1, and the flow depth decreases over the bump. The upstream, over the bump, and critical specific energy are

$$E_{s1} = y_1 + \frac{V_1^2}{2g} = (1.80\ \mathrm{m}) + \frac{(1.25\ \mathrm{m/s})^2}{2(9.81\ \mathrm{m/s^2})} = 1.88\ \mathrm{m}$$

The flow depth over the bump can be determined from

$$y_2^3 - (E_{s1} - \Delta z_b)y_2^2 + \frac{V_1^2}{2g}y_1^2 = 0 \quad \rightarrow \quad y_2^3 - (1.88 - 0.20\ \mathrm{m})y_2^2 + \frac{(1.25\ \mathrm{m/s})^2}{2(9.81\ \mathrm{m/s^2})}(1.80\ \mathrm{m})^2 = 0$$

Using an equation solver, the physically meaningful root of this equation is determined to be $y_2 =$ **1.576 m**. Then,

$$V_2 = \frac{y_1}{y_2}V_1 = \frac{1.8\ \mathrm{m}}{1.576\ \mathrm{m}}(1.25\ \mathrm{m/s}) = \mathbf{1.43\ m/s}$$

$$\mathrm{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{1.428\ \mathrm{m/s}}{\sqrt{(9.81\ \mathrm{m/s^2})(1.576\ \mathrm{m})}} = \mathbf{0.363}$$

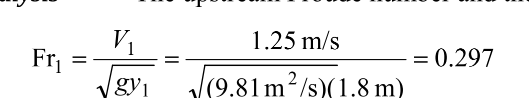
***Discussion*** The actual values may be somewhat different than those given above because of the frictional effects that are neglected in the analysis.

## 13-132

**Solution** Water flowing in a horizontal open channel encounters a bump. The bump height for which the flow over the bump is critical is to be determined.

***Assumptions*** **1** The flow is steady. **2** Frictional effects are negligible so that there is no dissipation of mechanical energy. **3** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The upstream Froude number and the critical depth are



$$\mathrm{Fr}_1 = \frac{V_1}{\sqrt{gy_1}} = \frac{1.25\ \mathrm{m/s}}{\sqrt{(9.81\ \mathrm{m^2/s})(1.8\ \mathrm{m})}} = 0.297$$

$$y_c = \left(\frac{\dot{V}^2}{gb^2}\right)^{1/3} = \left(\frac{(by_1V_1)^2}{gb^2}\right)^{1/3} = \left(\frac{y_1^2V_1^2}{g}\right)^{1/3} = \left(\frac{(1.8\ \mathrm{m})^2(1.25\ \mathrm{m/s})^2}{9.81\ \mathrm{m/s^2}}\right)^{1/3} = 0.802\ \mathrm{m}$$

The upstream flow is subcritical since Fr < 1, and the flow depth decreases over the bump. The upstream specific energy is

$$E_{s1} = y_1 + \frac{V_1^2}{2g} = (1.80\ \mathrm{m}) + \frac{(1.25\ \mathrm{m/s})^2}{2(9.81\ \mathrm{m/s^2})} = 1.88\ \mathrm{m}$$

Noting that the flow over the bump is critical and that $E_{s2} = E_{s1} - \Delta z_b$,

$$E_{s2} = E_c = \frac{3}{2}y_c = \frac{3}{2}(0.802\ \mathrm{m}) = 1.20\ \mathrm{m}$$

and

$$\Delta z_b = E_{s1} - E_{s2} = 1.88 - 1.20 = \mathbf{0.68\ m}$$

***Discussion*** If a higher bump is used, the flow will remain critical but the flow rate will decrease (the choking effect).

---

## 13-133

**Solution** Water flow through a wide rectangular channel undergoing a hydraulic jump is considered. It is to be shown that the ratio of the Froude numbers before and after the jump can be expressed in terms of flow depths $y_1$ and $y_2$ before and after the jump, respectively, as $\mathrm{Fr}_1/\mathrm{Fr}_2 = \sqrt{(y_2/y_1)^3}$ .

***Assumptions*** **1** The flow is steady. **2** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The Froude number for a wide channel of width $b$ and flow depth $y$ is given as

$$\mathrm{Fr} = \frac{V}{\sqrt{gy}} = \frac{\dot{V}/by}{\sqrt{gy}} = \frac{\dot{V}}{by\sqrt{gy}} = \frac{\dot{V}}{b\sqrt{gy^3}}$$

Expressing the Froude number before and after the jump and taking their ratio gives

$$\frac{\mathrm{Fr}_1}{\mathrm{Fr}_2} = \frac{\dot{V}/\left(b\sqrt{gy_1^3}\right)}{\dot{V}/\left(b\sqrt{gy_2^3}\right)} = \frac{\sqrt{gy_2^3}}{\sqrt{gy_1^3}} = \sqrt{\left(\frac{y_2}{y_1}\right)^3}$$

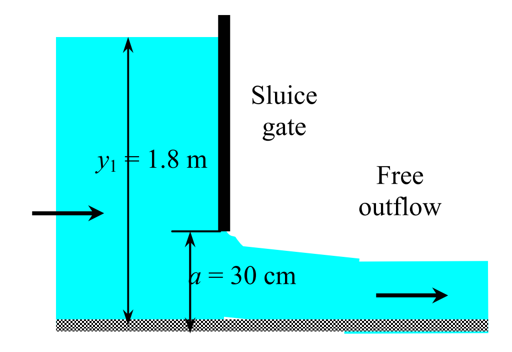
which is the desired result.

***Discussion*** Using the momentum equation, other relations such as $y_2 = 0.5y_1\left(-1+\sqrt{1+8\mathrm{Fr}_1^2}\right)$ can also be developed.

---

**13-134**

**Solution** A sluice gate with free outflow is used to control the flow rate of water. For specified flow depths, the flow rate per unit width and the downstream flow depth and velocity are to be determined.




***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** For free outflow, we only need the depth ratio $y_1/a$ to determine the discharge coefficient (for drowned outflow, we also need to know $y_2/a$ and thus the flow depth $y_2$ downstream the gate),

$$\frac{y_1}{a} = \frac{1.8\text{ m}}{0.30\text{ m}} = 6$$

The corresponding discharge coefficient is determined from Fig. 13-38 to be $C_d = 0.57$. Then the discharge rate per m width becomes

$$\dot{V} = C_d ba\sqrt{2gy_1} = 0.57\,(1\text{ m})(0.30\text{ m})\sqrt{2\,(9.81\text{ m/s}^2)(1.8\text{ m})} = \mathbf{1.02\ m^3/s}$$

The specific energy of a fluid remains constant during horizontal flow when the frictional effects are negligible, $E_{s1} = E_{s2}$. With these approximations, the flow depth and velocity past the gate become

$$E_{s1} = y_1 + \frac{V_1^2}{2g} = y_1 + \frac{\dot{V}^2}{2g(by_1)^2} = 1.8\text{ m} + \frac{(1.02\text{ m/s}^2)^2}{2(9.81\text{ m/s}^2)[(1\text{ m})(1.8\text{ m})]^2} = 1.816\text{ m}$$

$$E_{s2} = y_2 + \frac{V_2^2}{2g} = y_2 + \frac{\dot{V}^2}{2g(by_2)^2} = E_{s1} \rightarrow \ y_2 + \frac{(1.02\text{ m/s}^2)^2}{2(9.81\text{ m/s}^2)[(1\text{ m})(y_2)]^2} = 1.816\text{ m}$$

It gives $y_2$ = **0.179** m for flow depth as the physically meaningful root (positive and less than 1.8 m). Also,

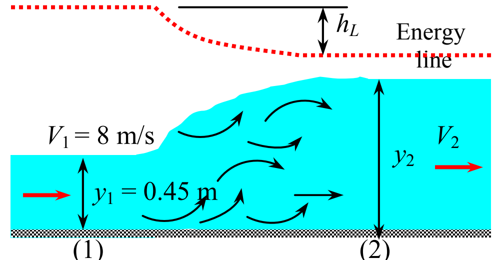
$$V_2 = \frac{\dot{V}}{A_c} = \frac{\dot{V}}{by_2} = \frac{1.02\text{ m}^3/\text{s}}{(1\text{ m})(0.179\text{ m})} = \mathbf{5.67\ m/s}$$

***Discussion*** In actual gates some frictional losses are unavoidable, and thus the actual velocity downstream will be lower.

**13-135**

**Solution** Water at a specified depth and velocity undergoes a hydraulic jump. The fraction of mechanical energy dissipated is to be determined.

***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible. **3** The channel is horizontal.




***Analysis*** The Froude number before the hydraulic jump is

$$\mathrm{Fr}_1 = \frac{V_1}{\sqrt{gy_1}} = \frac{8\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(0.45\text{ m})}} = 3.808$$

which is greater than 1. Therefore, the flow is indeed supercritical before the jump. The flow depth, velocity, and Froude number after the jump are

$$y_2 = 0.5y_1\left(-1+\sqrt{1+8\mathrm{Fr}_1^2}\right) = 0.5(0.45\text{ m})\left(-1+\sqrt{1+8\times 3.808^2}\right) = 2.209\text{ m}$$

$$V_2 = \frac{y_1}{y_2}V_1 = \frac{0.45\text{ m}}{2.209\text{ m}}(8\text{ m/s}) = 1.630\text{ m/s} \qquad \mathrm{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{1.630\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(2.209\text{ m})}} = 0.350$$

The head loss and the fraction of mechanical energy dissipated during the jump are

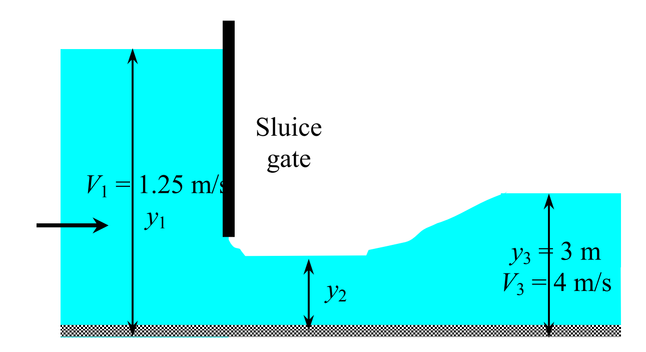
$$h_L = y_1 - y_2 + \frac{V_1^2 - V_2^2}{2g} = (0.45\text{ m}) - (2.209\text{ m}) + \frac{(8\text{ m/s})^2 - (1.63\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 1.368\text{ m}$$

$$\text{Dissipation ratio} = \frac{h_L}{E_{s1}} = \frac{h_L}{y_1(1+\mathrm{Fr}_1^2/2)} = \frac{1.368\text{ m}}{(0.45\text{ m})(1+3.808^2/2)} = \mathbf{0.369}$$

***Discussion*** Note that almost over one-third of the mechanical energy of the fluid is dissipated during hydraulic jump.

**13-136**

**Solution** The flow depth and average velocity of water after a hydraulic jump together with approach velocity to sluice gate are given. The flow rate per m width, the flow depths before and after the gate, and the energy dissipation ratio are to be determined.




***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The flow rate per m width of channel, flow depth before the sluice gate, and the Froude number after the jump is

$$\dot{V} = V_3 A_{c3} = V_3 b y_3 = (4\text{ m/s})(1\text{ m})(3\text{ m}) = \mathbf{12\ m^3/s}$$

$$y_1 = \frac{V_3}{V_1} y_3 = \frac{4\text{ m/s}}{1.25\text{ m/s}}(3\text{ m}) = \mathbf{9.60\ m}$$

$$\text{Fr}_3 = \frac{V_3}{\sqrt{g y_3}} = \frac{4\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(3\text{ m})}} = 0.7373$$

The flow dept, velocity, and Froude number before the jump are

$$y_2 = 0.5 y_3\left(-1+\sqrt{1+8\text{Fr}_3^2}\right) = 0.5(3\text{ m})\left(-1+\sqrt{1+8\times 0.7373^2}\right) = 1.969\text{ m} \cong \mathbf{1.97\ m}$$

$$V_2 = \frac{y_3}{y_2} V_3 = \frac{3\text{ m}}{1.969\text{ m}}(4\text{ m/s}) = 6.094\text{ m/s}$$

$$\text{Fr}_2 = \frac{V_2}{\sqrt{g y_2}} = \frac{6.094\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(1.969\text{ m})}} = 1.387$$

which is greater than 1, and thus the flow before the jump is indeed supercritical. The head loss and the fraction of mechanical energy dissipated during hydraulic jump are

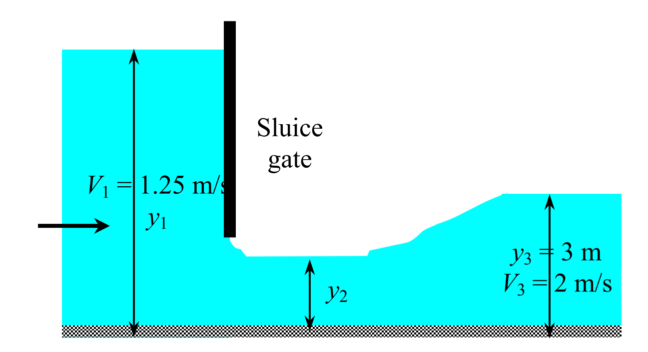
$$h_L = y_2 - y_3 + \frac{V_2^2 - V_3^2}{2g} = (1.969\text{ m}) - (3\text{ m}) + \frac{(6.094\text{ m/s})^2 - (4\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 0.0463\text{ m}$$

$$\text{Dissipation ratio} = \frac{h_L}{E_{s2}} = \frac{h_L}{y_2(1+\text{Fr}_2^2/2)} = \frac{0.0463\text{ m}}{(1.969\text{ m})(1+1.387^2/2)} = \mathbf{0.0120}$$

***Discussion*** Note that this is a "mild" hydraulic jump, and only 1.2% of the mechanical energy is wasted.

**13-137**

**Solution** The flow depth and average velocity of water after a hydraulic jump together with approach velocity to sluice gate are given. The flow rate per m width, the flow depths before and after the gate, and the energy dissipation ratio are to be determined.




***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The flow rate per m width of channel, flow depth before the sluice gate, and the Froude number after the jump is

$$\dot{V} = V_3 A_{c3} = V_3 b y_3 = (2 \text{ m/s})(1 \text{ m})(3 \text{ m}) = \mathbf{6\ m^3/s}$$

$$y_1 = \frac{V_3}{V_1} y_3 = \frac{2 \text{ m/s}}{1.25 \text{ m/s}}(3 \text{ m}) = \mathbf{4.8\ m}$$

$$\text{Fr}_3 = \frac{V_3}{\sqrt{g y_3}} = \frac{2 \text{ m/s}}{\sqrt{(9.81 \text{ m/s}^2)(3 \text{ m})}} = 0.3687$$

The flow dept, velocity, and Froude number before the jump are

$$y_2 = 0.5 y_3 \left(-1 + \sqrt{1 + 8\text{Fr}_3^2}\right) = 0.5(3 \text{ m})\left(-1 + \sqrt{1 + 8 \times 0.3687^2}\right) = 0.6671 \text{ m} \cong \mathbf{0.667\ m}$$

$$V_2 = \frac{y_3}{y_2} V_3 = \frac{3 \text{ m}}{0.6671 \text{ m}}(2 \text{ m/s}) = 8.994 \text{ m/s}$$

$$\text{Fr}_2 = \frac{V_2}{\sqrt{g y_2}} = \frac{8.994 \text{ m/s}}{\sqrt{(9.81 \text{ m/s}^2)(0.6671 \text{ m})}} = 3.516$$

which is greater than 1, and thus the flow before the jump is indeed supercritical. The head loss and the fraction of mechanical energy dissipated during hydraulic jump are

$$h_L = y_2 - y_3 + \frac{V_2^2 - V_3^2}{2g} = (0.6671 \text{ m}) - (3 \text{ m}) + \frac{(8.994 \text{ m/s})^2 - (2 \text{ m/s})^2}{2(9.81 \text{ m/s}^2)} = 1.586 \text{ m}$$

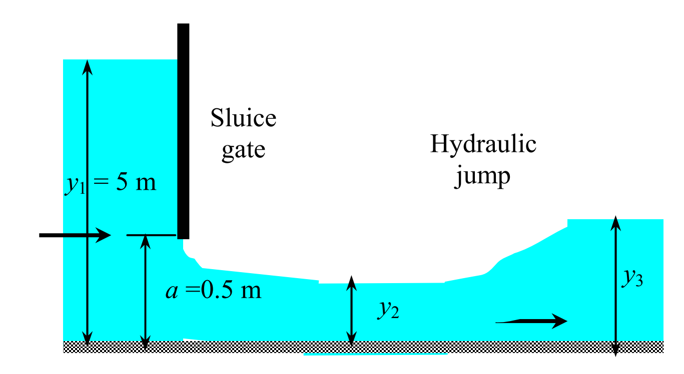
$$\text{Dissipation ratio} = \frac{h_L}{E_{s2}} = \frac{h_L}{y_2(1 + \text{Fr}_2^2/2)} = \frac{1.586 \text{ m}}{(0.6671 \text{ m})(1 + 3.516^2/2)} = \mathbf{0.331}$$

***Discussion*** Note that this is a fairly "strong" hydraulic jump, wasting 33.1% of the mechanical energy of the fluid.

**13-138**

**Solution** Water from a lake is discharged through a sluice gate into a channel where uniform flow conditions are established, and then undergoes a hydraulic jump. The flow depth, velocity, and Froude number after the jump are to be determined.

***Assumptions*** **1** The flow is steady. **2** The channel is sufficiently wide so that the end effects are negligible. **3** The effects of channel slope on hydraulic jump are negligible.




***Properties*** The Manning coefficient for an open channel made of finished concrete is $n = 0.012$ (Table 13-1).

***Analysis*** For free outflow, we only need the depth ratio $y_1/a$ to determine the discharge coefficient,

$$\frac{y_1}{a} = \frac{5\text{ m}}{0.5\text{ m}} = 10$$

The corresponding discharge coefficient is determined from Fig. 13-38 to be $C_d = 0.58$. Then the discharge rate per m width ($b = 1$ m) becomes

$$\dot{V} = C_d ba\sqrt{2gy_1} = 0.58\,(1\text{ m})(0.5\text{ m})\sqrt{2\,(9.81\text{ m/s}^2)(5\text{ m})} = 2.872\text{ m}^3\text{/s}$$

For wide channels, hydraulic radius is the flow depth and thus $R_h = y_2$. Then the flow depth in uniform flow after the gate is determined from the Manning's equation to be

$$\dot{V} = \frac{a}{n} A_c R_h^{2/3} S_0^{1/2} \quad \rightarrow \quad 2.872\text{ m}^3\text{/s} = \frac{1\text{ m}^{1/3}/s}{0.012}[(1\text{ m})y_2](y_2)^{2/3}\,0.004^{1/2}$$

It gives $y_2 = 0.6948$ m, which is also the flow depth before water undergoes a hydraulic jump. The flow velocity and Froude number in uniform flow are

$$V_2 = \frac{\dot{V}}{b\,y_2} = \frac{2.872\text{ m}^3\text{/s}}{(1\text{m})\,(0.6948\text{ m})} = 4.134\text{ m/s}$$

$$\text{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{4.134\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(0.6948\text{ m})}} = 1.584$$

Then the flow depth, velocity, and Froude number after the jump (state 3) become

$$y_3 = 0.5y_2\left(-1+\sqrt{1+8\text{Fr}_2^2}\right) = 0.5(0.6948\text{ m})\left(-1+\sqrt{1+8\times1.584^2}\right) = \mathbf{1.25\text{ m}}$$

$$V_3 = \frac{y_2}{y_3}V_2 = \frac{0.6948\text{ m}}{1.25\text{ m}}(4.134\text{ m/s}) = \mathbf{2.30\text{ m/s}}$$

$$\text{Fr}_3 = \frac{V_3}{\sqrt{gy_{23}}} = \frac{2.30\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(1.25\text{ m})}} = \mathbf{0.659}$$

***Discussion*** This is a relatively "mild" jump. It can be shown that the head loss during hydraulic jump is 0.049 m, which corresponds to an energy dissipation ratio of 3.1%.

**13-139**

**Solution** Water is discharged from a dam into a wide spillway to reduce the risk of flooding by dissipating a large fraction of mechanical energy via hydraulic jump. For specified flow depths, the velocities before and after the jump, and the mechanical power dissipated per meter with of the spillway are to be determined.

***Assumptions*** **1** The flow is steady or quasi-steady. **2** The channel is sufficiently wide so that the end effects are negligible.

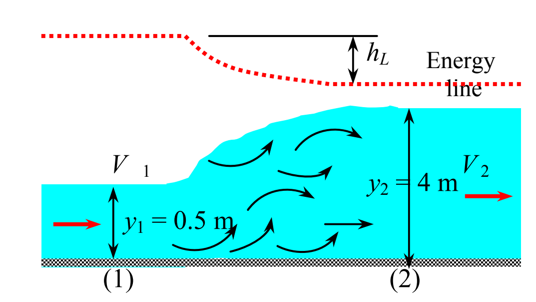
***Properties*** The density of water is 1000 kg/m$^3$.

***Analysis*** The Froude number and velocity before the jump are

$$\frac{y_2}{y_1} = 0.5\left(-1+\sqrt{1+8\text{Fr}_1^2}\right) \rightarrow \frac{4\text{ m}}{0.5\text{ m}} = 0.5\left(-1+\sqrt{1+8\text{Fr}_1^2}\right)$$

which gives $\text{Fr}_1 = 6$. Also, from the definition of Froude number,

$$V_1 = \text{Fr}_1\sqrt{gy_1} = (6)\sqrt{(9.81\text{ m/s}^2)(0.5\text{ m})} = \textbf{13.3 m/s}$$




Velocity and Froude number after the jump are

$$V_2 = \frac{y_1}{y_2}V_1 = \frac{0.5\text{ m}}{4\text{ m}}(13.3\text{ m/s}) = \textbf{1.66 m/s}$$

$$\text{Fr}_2 = \frac{V_2}{\sqrt{gy_2}} = \frac{1.66\text{ m/s}}{\sqrt{(9.81\text{ m/s}^2)(4\text{ m})}} = 0.265$$

The head loss is determined from the energy equation to be

$$h_L = y_1 - y_2 + \frac{V_1^2 - V_2^2}{2g} = (0.5\text{ m}) - (4\text{ m}) + \frac{(13.3\text{ m/s})^2 - (1.66\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 5.36\text{ m}$$

The volume and mass flow rates of water per m width are

$$\dot{V} = V_1 A_{c1} = V_1 b y_1 = (13.3\text{ m/s})(1\text{ m})(0.5\text{ m}) = 6.64\text{ m}^3\text{/s}$$

$$\dot{m} = \rho\dot{V} = (1000\text{ kg/m}^3)(6.64\text{ m}^3\text{/s}) = 6640\text{ kg/s}$$

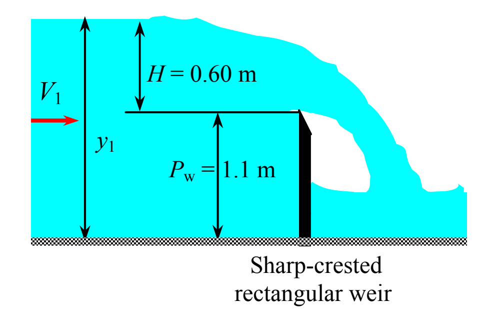
Then the dissipated mechanical power becomes

$$\dot{E}_{\text{dissipated}} = \dot{m}gh_L = (6640\text{ kg/s})(9.81\text{ m/s}^2)(5.36\text{ m})\left(\frac{1\text{ kN}}{1000\text{ kg}\cdot\text{m/s}^2}\right) = 349\text{ kNm/s} = \textbf{349 kW}$$

***Discussion*** The results show that the hydraulic jump is a highly dissipative process, wasting 349 kW of power in this case.

**13-140**

**Solution** The flow rate in an open channel is to be measured using a sharp-crested rectangular weir. For a measured value of flow depth upstream, the flow rate is to be determined.




***Assumptions*** **1** The flow is steady. **2** The upstream velocity head is negligible. **3** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The weir head is given to be $H$ = 0.60 m. The discharge coefficient of the weir is

$$C_{wd,\text{rec}} = 0.598 + 0.0897\frac{H}{P_w} = 0.598 + 0.0897\frac{0.60\text{ m}}{1.1\text{ m}} = 0.6469$$

The condition $H/P_w < 2$ is satisfied since 0.60/1.1 = 0.55. Then the water flow rate through the channel becomes

$$\dot{V} = C_{wd,\text{rec}}\frac{2}{3}b\sqrt{2g}H^{3/2}$$
$$= (0.6469)\frac{2}{3}(6\text{ m})\sqrt{2(9.81\text{ m/s}^2)}(0.60\text{ m})^{3/2}$$
$$= \mathbf{5.33\ m^3/s}$$

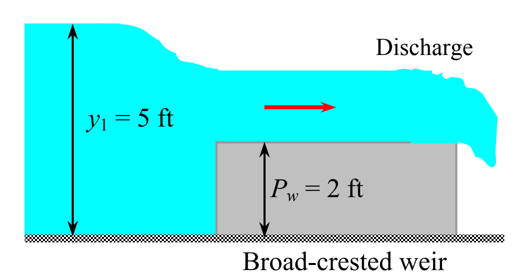
***Discussion*** The upstream velocity and the upstream velocity head are

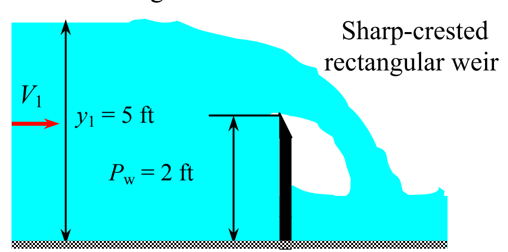
$$V_1 = \frac{\dot{V}}{by_1} = \frac{5.33\text{ m}^3/\text{s}}{(6\text{ m})(1.70\text{ m})} = 0.522\text{ m/s} \quad \text{and} \quad \frac{V_1^2}{2g} = \frac{(0.522\text{ m/s})^2}{2(9.81\text{ m/s}^2)} = 0.014\text{ m}$$

This is 2.3% of the weir head, which is negligible. When the upstream velocity head is considered, the flow rate becomes 5.50 $m^3/s$, which is about 3 percent higher than the value determined above. Therefore, the assumption of negligible velocity head is reasonable in this case.

**13-141E**

**Solution** The flow rates in two open channels are to be measured using a sharp-crested weir in one and a broad-crested rectangular weir in the other. For identical flow depths, the flow rates through both channels are to be determined.




***Assumptions*** **1** The flow is steady. **2** The upstream velocity head is negligible. **3** The channel is sufficiently wide so that the end effects are negligible.

***Analysis*** The weir head is

$$H = y_1 - P_w = 5.0\text{ ft} - 2.0\text{ ft} = 3.0\text{ ft}$$

The condition $H/P_w < 2$ is satisfied since 3.0/2.0 = 1.5. The discharge coefficients of the weirs are

**Sharp-crested weir:**

$$C_{wd,\text{sharp}} = 0.598 + 0.0897\frac{H}{P_w} = 0.598 + 0.0897\frac{3.0\text{ ft}}{2.0\text{ ft}} = 0.7326$$

$$\dot{V}_{\text{sharp}} = C_{wd,sharp}\frac{2}{3}b\sqrt{2g}H^{3/2} = (0.7326)\frac{2}{3}(12\text{ ft})\sqrt{2(32.2\text{ ft/s}^2)}(3.0\text{ ft})^{3/2} = \mathbf{244\ m^3/s}$$

**Broad-crested weir:**

$$C_{wd,\text{broad}} = \frac{0.65}{\sqrt{1+H/P_w}} = \frac{0.65}{\sqrt{1+(3.0\text{ ft})/(2.0\text{ ft})}} = 0.4111$$

$$\dot{V}_{\text{broad}} = C_{wd,\text{broad}}b\sqrt{g}\left(\frac{2}{3}\right)^{3/2}H^{3/2} = (0.4111)(12\text{ ft})(2/3)^{3/2}\sqrt{32.2\text{ ft/s}^2}(3.0\text{ ft})^{3/2} = \mathbf{79.2\ m^3/s}$$

***Discussion*** Note that the flow rate in the channel with the broad-crested weir is much less than the channel with the sharp-crested weir. Also, if the upstream velocity is taken into consideration, the flow rate would be 270 $ft^3/s$ (11% difference) for the channel with the sharp-crested weir, and 80.3 $ft^3/s$ (1% difference) for the one with broad-crested weir. Therefore, the assumption of negligible dynamic head is not quite appropriate for the channel with the sharp-crested weir.

---

## Design and Essay Problems

**13-142 to 13-143**

**Solution** Students' essays and designs should be unique and will differ from each other.

---



# Solutions Manual for

# Fluid Mechanics: Fundamentals and Applications

# by Çengel & Cimbala

---

# CHAPTER 14
# TURBOMACHINERY

---

## General Problems

### 14-1C

**Solution** We are to discuss energy producing and energy absorbing devices.

***Analysis*** A more common term for an energy producing turbomachine is a **turbine**. **Turbines extract energy from the moving fluid, and convert that energy into useful mechanical energy in the surroundings**, usually in the form of a rotating shaft. Thus, the phrase "energy producing" is from a frame of reference of the fluid – the fluid loses energy as it drives the turbine, producing energy to the surroundings. On the other hand, a more common term for an energy absorbing turbomachine is a **pump**. **Pumps absorb mechanical energy from the surroundings, usually in the form of a rotating shaft, and increase the energy of the moving fluid**. Thus, the phrase "energy absorbing" is from a frame of reference of the fluid – the fluid gains or absorbs energy as it flows through the pump.

***Discussion*** From the frame of reference of the surroundings, a pump absorbs energy from the surroundings, while a turbine produces energy to the surroundings. Thus, you may argue that the terminology also holds for the frame of reference of the surroundings. This alternative explanation is also acceptable.

### 14-2C

**Solution** We are to discuss the differences between fans, blowers, and compressors.

***Analysis*** A *fan* is a gas pump with relatively l**ow pressure rise** and **high flow rate**. A ***blower*** is a gas pump with relatively **moderate to high pressure rise** and **moderate to high flow rate**. A ***compressor*** is a gas pump designed to deliver a very **high pressure rise**, typically at **low to moderate flow rates**.

***Discussion*** The boundaries between these three types of pump are not always clearly defined.

### 14-3C

**Solution** We are to list examples of fans, blowers, and compressors.

***Analysis*** Common examples of fans are **window fans, ceiling fans, fans in computers and other electronics equipment, radiator fans in cars**, etc. Common examples of blowers are **leaf blowers, hair dryers, air blowers in furnaces and automobile ventilation systems**. Common examples of compressors are **tire pumps, refrigerator and air conditioner compressors**.

***Discussion*** Students should come up with a diverse variety of examples.

### 14-4C

**Solution** We are to discuss the difference between a positive-displacement turbomachine and a dynamic turbomachine.

***Analysis*** **A *positive-displacement turbomachine* is a device that contains a closed volume; energy is transferred to the fluid (pump) or from the fluid (turbine) via movement of the boundaries of the closed volume**. On the other hand, **a *dynamic turbomachine* has no closed volume; instead, energy is transferred to the fluid (pump) or from the fluid (turbine) via rotating blades**. Examples of positive-displacement pumps include well **pumps, hearts, some aquarium pumps, and pumps designed to release precise volumes of medicine**. Examples of positive-displacement turbines include **water meters and gas meters in the home**. Examples of dynamic pumps include **fans, centrifugal blowers, airplane propellers, centrifugal water pumps** (like in a car engine), etc. Examples of dynamic turbines include **windmills, wind turbines, turbine flow meters**, etc.

***Discussion*** Students should come up with a diverse variety of examples.

**14-5C**

**Solution** We are to discuss the difference between brake horsepower and water horsepower, and then discuss pump efficiency.

***Analysis*** In turbomachinery terminology, **brake horsepower is the power actually delivered to the pump through the shaft**. (One may also call it "shaft power".) On the other hand, **water horsepower is the useful portion of the brake horsepower that is actually delivered to the fluid**. Water horsepower is always less than brake horsepower; hence **pump efficiency is defined as the ratio of water horsepower to brake horsepower**.

***Discussion*** For a turbine, efficiency is defined in the opposite way, since brake horsepower is *less* than water horsepower.

---

**14-6C**

**Solution** We are to discuss the difference between brake horsepower and water horsepower, and then discuss turbine efficiency.

***Analysis*** In turbomachinery terminology, **brake horsepower is the power actually delivered by the turbine to the shaft.** (One may also call it "shaft power".) On the other hand, **water horsepower is the power extracted from the water flowing through the turbine**. Water horsepower is always greater than brake horsepower; because of inefficiencies; hence **turbine efficiency is defined as the ratio of brake horsepower to water horsepower**.

***Discussion*** For a pump, efficiency is defined in the opposite way, since brake horsepower is *greater* than water horsepower.

---

**14-7C**

**Solution** We are to explain the "extra" term in the Bernoulli equation in a rotating reference frame.

***Analysis*** A rotating reference frame is not an inertial reference frame. When we move outward in the radial direction, the absolute velocity at this location is faster due to the rotating body, since $v_\theta$ is equal to $\omega r$. When solving a turbomachinery problem in a rotating reference frame, we use the *relative* fluid velocity (velocity relative to the rotating reference frame). Thus, **in order for the Bernoulli equation to be physically correct, we must subtract the absolute velocity of the rotating body so that the equation applies to an inertial reference frame. This accounts for the "extra" term**.

***Discussion*** The Bernoulli equation is the same physical equation in either the absolute or the rotating reference frame, but it is more convenient to use the form with the extra term in turbomachinery applications.

---

## 14-8

**Solution** We are to determine how the average speed at the outlet compares to the average speed at the inlet of a water pump.

***Assumptions*** **1** The flow is steady (in the mean). **2** The water is incompressible.

***Analysis*** Conservation of mass requires that the mass flow rate in equals the mass flow rate out. Thus,

*Conservation of mass*: $$\dot{m}_{\text{in}} = \rho_{\text{in}} V_{\text{in}} A_{\text{in}} = \dot{m}_{\text{out}} = \rho_{\text{out}} V_{\text{out}} A_{\text{out}}$$

or, since the cross-sectional area is proportional to the square of diameter,

$$V_{\text{out}} = V_{\text{in}} \frac{\rho_{\text{in}}}{\rho_{\text{out}}} \left( \frac{D_{\text{in}}}{D_{\text{out}}} \right)^2 = V_{\text{in}} \left( \frac{D_{\text{in}}}{D_{\text{out}}} \right)^2 \qquad (1)$$

(a) For the case where $D_{\text{out}} < D_{\text{in}}$, $V_{\text{out}}$ must be **greater than** $V_{\text{in}}$.

(b) For the case where $D_{\text{out}} = D_{\text{in}}$, $V_{\text{out}}$ must be **equal to** $V_{\text{in}}$.

(c) For the case where $D_{\text{out}} > D_{\text{in}}$, $V_{\text{out}}$ must be **less than** $V_{\text{in}}$.

***Discussion*** A pump does not necessarily increase the speed of the fluid passing through it. In fact, the average speed through the pump can actually *decrease*, as it does here in part (c).

---

## 14-9

**Solution** For an air compressor with equal inlet and outlet areas, and with both density and pressure increasing, we are to determine how the average speed at the outlet compares to the average speed at the inlet.

***Assumptions*** **1** The flow is steady.

***Analysis*** Conservation of mass requires that the mass flow rate in equals the mass flow rate out. The cross-sectional areas of the inlet and outlet are the same. Thus,

*Conservation of mass*: $$\dot{m}_{\text{in}} = \rho_{\text{in}} V_{\text{in}} A_{\text{in}} = \dot{m}_{\text{out}} = \rho_{\text{out}} V_{\text{out}} A_{\text{out}}$$

or

$$V_{\text{out}} = V_{\text{in}} \frac{\rho_{\text{in}}}{\rho_{\text{out}}} \qquad (1)$$

Since $\rho_{\text{in}} < \rho_{\text{out}}$, $V_{\text{out}}$ must be **less than** $V_{\text{in}}$.

***Discussion*** A compressor does not necessarily increase the speed of the fluid passing through it. In fact, the average speed through the pump can actually *decrease*, as it does here.

---

## Pumps

### 14-10C
**Solution** We are to list and define the three categories of dynamic pumps.

***Analysis*** The three categories are: **Centrifugal flow pump –** fluid enters axially (in the same direction as the axis of the rotating shaft) in the center of the pump, but is discharged radially (or tangentially) along the outer radius of the pump casing. **Axial-flow pump** – fluid enters and leaves axially, typically only along the outer portion of the pump because of blockage by the shaft, motor, hub, etc. **Mixed-flow pump** – intermediate between centrifugal and axial, with the flow entering axially, not necessarily in the center, but leaving at some angle between radially and axially.

***Discussion*** There are also some non-rotary dynamic pumps, such as jet pumps and electromagnetic pumps, that are not discussed in this text.

### 14-11C
**Solution**

(*a*) *False*: Actually, backward-inclined blades yield the highest efficiency.
(*b*) *True*: The pressure rise is higher, but at the cost of less efficiency.
(*c*) *True*: In fact, this is the primary reason for choosing forward-inclined blades.
(*d*) *False*: Actually, the opposite is true – a pump with forward-inclined blades usually has more blades, but they are usually smaller.

### 14-12C
**Solution** We are to choose which pump location is better and explain why.

***Analysis*** The two systems are identical except for the location of the pump (and some minor differences in pipe layout). The overall length of pipe, number of elbows, elevation difference between the two reservoir free surfaces, etc. are the same. **Option (a) is better because it has the pump at a lower elevation, increasing the net positive suction head, and lowering the possibility of pump cavitation**. In addition, the length of pipe from the lower reservoir to the pump inlet is smaller in Option (a), and there is one less elbow between the lower reservoir and the pump inlet, thereby decreasing the head loss upstream of the pump – both of which also increase NPSH, and reduce the likelihood of cavitation.

***Discussion*** Another point is that if the pump is not self-priming, Option (b) may run into start-up problems if the free surface of the lower reservoir falls below the elevation of the pump inlet. Since the pump in Option (a) is below the reservoir, self-priming is not an issue.

### 14-13C
**Solution** We are to define and discuss NPSH and $\text{NPSH}_{\text{required}}$.

***Analysis*** Net positive suction head (NPSH) is defined as **the difference between the pump's inlet stagnation pressure head and the vapor pressure head**,

$$\text{NPSH} = \left(\frac{P}{\rho g} + \frac{V^2}{2g}\right)_{\text{pump inlet}} - \frac{P_v}{\rho g}$$

We may think of NPSH as the actual or available net positive suction head. On the other hand, required net positive suction head ($\text{NPSH}_{\text{required}}$) is defined as the **minimum NPSH necessary to avoid cavitation in the pump**. As long as the actual NPSH is greater than $\text{NPSH}_{\text{required}}$, there should be no cavitation in the pump.

***Discussion*** Although NPSH and $\text{NPSH}_{\text{required}}$ are measured at the pump inlet, cavitation (if present) happens somewhere inside the pump, typically on the suction surface of the rotating pump impeller blades.

**14-14C**

**Solution**

(*a*) *True*: As volume flow rate increases, not only does $NPSH_{required}$ increase, but the available NPSH decreases, increasing the likelihood that NPSH will drop below $NPSH_{required}$ and cause cavitation to occur.
(*b*) *False*: $NPSH_{required}$ is *not* a function of water temperature, although available NPSH does depend on water temperature.
(*c*) *False*: Available NPSH actually *decreases* with increasing water temperature, making cavitation more likely to occur.
(*d*) *False*: Actually, warmer water causes cavitation to be *more likely*. The best way to think about this is that warmer water is already closer to its boiling point, so cavitation is more likely to happen in warm water than in cold water.

---

**14-15C**

**Solution** We are to explain why dissimilar pumps should not be arranged in series or in parallel.

***Analysis*** Arranging dissimilar pumps in series can create problems because the volume flow rate through each pump must be the same, but the overall pressure rise is equal to the pressure rise of one pump plus that of the other. If the pumps have widely different performance curves, the smaller pump may be forced to operate beyond its free delivery flow rate, whereupon it acts like a head *loss*, reducing the total volume flow rate. Arranging dissimilar pumps in parallel can create problems because the overall pressure rise must be the same, but the net volume flow rate is the sum of that through each branch. If the pumps are not sized properly, the smaller pump may not be able to handle the large head imposed on it, and the flow in its branch could actually be *reversed*; this would inadvertently reduce the overall pressure rise. In either case, the power supplied to the smaller pump would be wasted.

***Discussion*** If the pumps are not significantly dissimilar, a series or parallel arrangement of the pumps might be wise.

---

**14-16C**

**Solution**

(*a*) *True*: The maximum volume flow rate occurs when the net head is zero, and this “free delivery” flow rate is typically much higher than that at the BEP.
(*b*) *True*: By definition, there is no flow rate at the shutoff head. Thus the pump is not doing any useful work, and the efficiency must be zero.
(*c*) *False*: Actually, the net head is typically greatest near the shutoff head, at zero volume flow rate, not near the BEP.
(*d*) *True*: By definition, there is no head at the pump’s free delivery. Thus, the pump is working against no “resistance”, and is therefore not doing any useful work, and the efficiency must be zero.

---

**14-17C**

**Solution** We are to discuss ways to improve the cavitation performance of a pump, based on the equation for NPSH.

***Analysis*** NPSH is defined as

$$\text{NPSH} = \left(\frac{P}{\rho g} + \frac{V^2}{2g}\right)_{\text{pump inlet}} - \frac{P_v}{\rho g} \quad (1)$$

To avoid cavitation, NPSH must be increased as much as possible. For a given liquid at a given temperature, the vapor pressure head (last term on the right side of Eq. 1) is constant. Hence, the only way to increase NPSH is to increase the stagnation pressure head at the pump inlet. We list several ways to increase the available NPSH: (1) **Lower the pump or raise the inlet reservoir level. (2) Use a larger diameter pipe upstream of the pump. (3) Re-route the piping system such that fewer minor losses (elbows, valves, etc.) are encountered upstream of the pump. (4) Shorten the length of pipe upstream of the pump. (5) Use a smoother pipe. (6) Use elbows, valves, inlets, etc. that have smaller minor loss coefficients**. Suggestion (1) raises NPSH by increasing the hydrostatic component of pressure at the pump inlet. Suggestions (2) through (6) raise NPSH by lowering the irreversible head losses, thereby increasing the pressure at the pump inlet.

***Discussion*** By definition, when the available NPSH falls below the required NPSH, the pump is prone to cavitation, which should be avoided if at all possible.

---

## 14-18C

**Solution**

(*a*) *False*: Since the pumps are in series, the volume flow rate through each pump must be the same: $\dot{V} = \dot{V}_1 = \dot{V}_2$ .

(*b*) *True*: The net head increases by $H_1$ through the first pump, and then by $H_2$ through the second pump. The overall rise in net head is thus the sum of the two.

(*c*) *True*: Since the pumps are in parallel, the total volume flow rate is the sum of the individual volume flow rates.

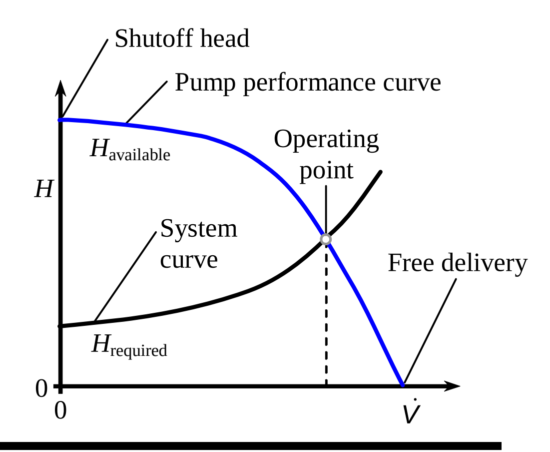
(*d*) *False*: For pumps in parallel, the change in pressure from the upstream junction to the downstream junction is the same regardless of which parallel branch is under consideration. Thus, even though the volume flow rate may not be the same in each branch, the net head must be the same: $H = H_1 = H_2$.

---

## 14-19C

**Solution** We are to label several items on the provided plot.

***Analysis*** The figure is re-drawn here, and the requested items are labeled.




***Discussion*** Also labeled are the available net head, corresponding to the pump performance curve, and the required net head, corresponding to the system curve. The intersection of these two curves is the operating point of the pump.

---

## 14-20

**Solution** We are to determine which free surface is at higher elevation, and justify our answer with the energy equation.

***Analysis*** It is simplest to consider zero-flow conditions ($\dot{V}$ = 0), at which we see that the required net head is positive. This implies that, even when there is no flow between the two tanks, the pump would need to provide some net head just to overcome the pressure differences. Since there is no flow, pressure differences can come only from gravity. Hence, **the outlet tank's free surface must be higher than that of the inlet tank**. Mathematically, we apply the energy equation in head form between the inlet tank's free surface (1) and the outlet tank's free surface (2),

*Energy equation at zero flow conditions*:

$$H_{\text{required}} = h_{\text{pump,u}} = \frac{P_2 - P_1}{\rho g} + \frac{\alpha_2 V_2^2 - \alpha_1 V_1^2}{2g} + (z_2 - z_1) + h_{\text{turbine}} + h_{L,\text{total}} \qquad (1)$$

Since both free surfaces are at atmospheric pressure, $P_1 = P_2 = P_{\text{atm}}$, and the first term on the right side of Eq. 1 vanishes. Furthermore, since there is no flow, $V_1 = V_2 = 0$, and the second term vanishes. There is no turbine in the control volume, so the second-to-last term is zero. Finally, there are no irreversible head losses since there is no flow, and the last term is also zero. Equation 1 reduces to

$$H_{\text{required}} = h_{\text{pump}} = (z_2 - z_1) \qquad (2)$$

Since $H_{\text{required}}$ is positive on Fig. P14-19 at $\dot{V}$ = 0, the quantity $(z_2 - z_1)$ must also be positive by Eq. 2. Thus we have shown mathematically that **the outlet tank's free surface is higher in elevation than that of the inlet tank**.

***Discussion*** If the reverse were true (outlet tank free surface lower than inlet tank free surface), $H_{\text{required}}$ at $\dot{V}$ = 0 would be *negative*, implying that the pump would need to supply enough *negative* net head to hold back the natural tendency of the water to flow from higher to lower elevation. In reality, the pump would not be able to do this unless it were spun backwards.

---

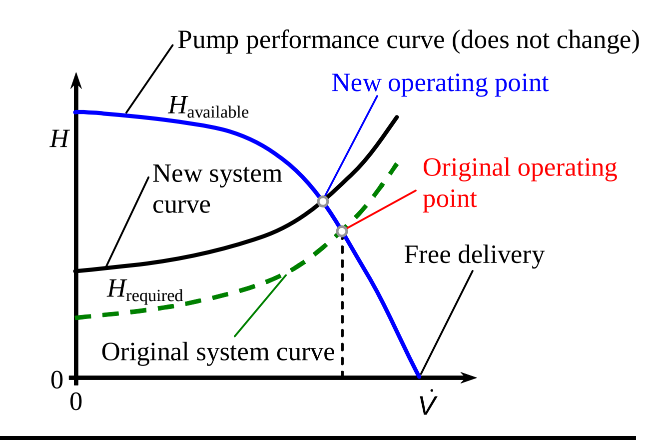
## 14-21

**Solution** We are to discuss what would happen to the pump performance curve, the system curve, and the operating point if the free surface of the outlet tank were raised to a higher elevation.

***Analysis*** The pump is the same pump regardless of the locations of the inlet and outlet tanks' free surfaces; thus, **the pump performance curve does not change**. The energy equation is

$$H_{\text{required}} = h_{\text{pump,u}} = \frac{P_2 - P_1}{\rho g} + \frac{\alpha_2 V_2^2 - \alpha_1 V_1^2}{2g} + (z_2 - z_1) + \cancel{h_{\text{turbine}}} + h_{L,\text{total}} \qquad (1)$$

Since the only thing that changes is the elevation difference, Eq. 1 shows that $H_{\text{required}}$ shifts up as $(z_2 - z_1)$ increases. Thus, **the system curve rises linearly with elevation increase**. A plot of $H$ versus $\dot{V}$ is plotted, and the new operating point is labeled. Because of the upward shift of the system curve, **the operating point moves to a lower value of volume flow rate**.




***Discussion*** The shift of operating point to lower $\dot{V}$ agrees with our physical intuition. Namely, as we raise the elevation of the outlet, the pump has to do more work to overcome gravity, and we expect the flow rate to decrease accordingly.

---

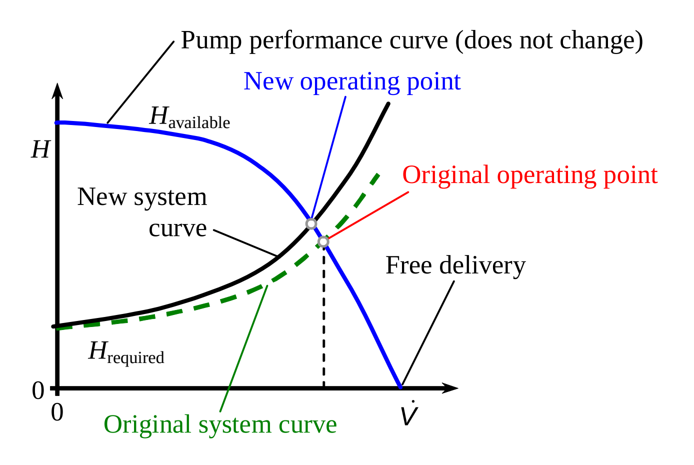
## 14-22

**Solution** We are to discuss what would happen to the pump performance curve, the system curve, and the operating point if a valve changes from 100% to 50% open.

***Analysis*** The pump is the same pump regardless of the locations of the inlet and outlet tanks' free surfaces; thus, **the pump performance curve does not change**. The energy equation is

$$H_{\text{required}} = h_{\text{pump,u}} = \frac{P_2 - P_1}{\rho g} + \frac{\alpha_2 V_2^2 - \alpha_1 V_1^2}{2g} + (z_2 - z_1) + \cancel{h_{\text{turbine}}} + h_{L,\text{total}} \qquad (1)$$

Since both free surfaces are open to the atmosphere, the pressure term vanishes. Since both $V_1$ and $V_2$ are negligibly small at the free surface (the tanks are large), the second term on the right also vanishes. The elevation difference ($z_2 - z_1$) does not change, and so the only term in Eq. 1 that is changed by closing the valve is the irreversible head loss term. We know that the minor loss associated with a valve increases significantly as the valve is closed. Thus, **the system curve** (the curve of $H_{\text{required}}$ versus $\dot{V}$) **increases more rapidly with volume flow rate (has a larger slope) when the valve is partially closed**. A sketch of $H$ versus $\dot{V}$ is plotted, and the new operating point is labeled. Because of the higher system curve, ***the operating point moves to a lower value of volume flow rate***, as indicated on the figure. I.e., **the volume flow rate decreases**.

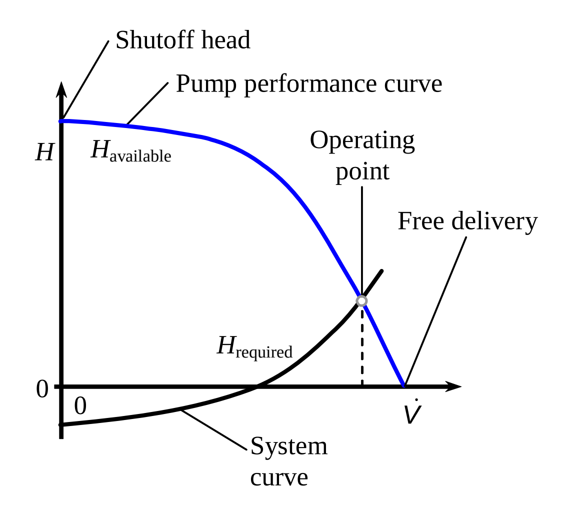



***Discussion*** The shift of operating point to lower $\dot{V}$ agrees with our physical intuition. Namely, as we close the valve somewhat, the pump has to do more work to overcome the losses, and we expect the flow rate to decrease accordingly.

---

**14-23**

**Solution** We are to create a qualitative plot of pump net head versus pump capacity.




***Analysis*** The result is shown in the figure, and the requested items are labeled. Also labeled are the available net head, corresponding to the pump performance curve, and the required net head, corresponding to the system curve. The intersection of these two curves is the operating point of the pump. Note that since the elevation of the outlet is lower than that of the free surface of the inlet tank, the required net head must be *negative* at zero flow rate conditions, as sketched, implying that the pump holds back the natural tendency of the water to flow from higher to lower elevation. Only at higher flow rates does the system curve rise to positive values of $H_{required}$.

***Discussion*** A real pump cannot produce negative net head at zero volume flow rate unless its blades are spun in the opposite direction than that for which they are designed.

**14-24**

**Solution** We are to estimate the volume flow rate through a piping system.

***Assumptions*** **1** Since the reservoir is large, the flow is nearly steady. **2** The water is incompressible. **3** The water is at room temperature. **4** The flow in the pipe is fully developed and turbulent, with $\alpha = 1.05$.

***Properties*** The density and viscosity of water at $T = 20^{\circ}$C are 998.0 kg/m$^3$ and $1.002 \times 10^{-3}$ kg/m·s respectively.

***Analysis*** By definition, at free delivery conditions, the net head across the pump is zero. Thus, there is no loss or gain of pressure across the pump, and we can essentially ignore it in the calculations here. We apply the head form of the steady energy equation from location 1 to location 2,

$$H_{\text{required}} = h_{\text{pump}} = 0 = \cancel{\frac{P_2 - P_1}{\rho g}} + \frac{\alpha_2 V_2^{\,2} - \cancel{V_1^{\,2}}}{2g} + \left(z_2 - z_1\right) + \cancel{h_{\text{turbine}}} + h_{L,\text{total}} \qquad (1)$$

where the pressure term vanishes since the free surface at location 1 and at the exit (location 2) are both open to the atmosphere. The inlet velocity term disappears since $V_1$ is negligibly small at the free surface. Thus, Eq. 1 reduces to a balance between supplied potential energy head ($z_1 - z_2$), kinetic energy head at the exit $\alpha_2 V_2^2/2g$, and irreversible head losses,

$$\left(z_1 - z_2\right) = \frac{\alpha_2 V_2^{\,2}}{2g} + h_{L,\text{total}} \qquad (2)$$

The total irreversible head loss in Eq. 2 consists of both major and minor losses. We split the minor losses into those associated with the mean velocity $V$ through the pipe, and the minor loss associated with the contraction, based on exit velocity $V_2$,

$$\left(z_1 - z_2\right) = \frac{\alpha_2 V_2^{\,2}}{2g} + \frac{V^2}{2g}\left(f\frac{L}{D} + \sum_{\text{pipe}} K_L\right) + \frac{V_2^{\,2}}{2g} K_{L,\text{contraction}} \qquad (3)$$

where $\sum_{\text{pipe}} K_L$ = 0.50 + 2(2.4) + 3(0.90) = 8.0, and $K_{L,\text{contraction}}$ = 0.15.

By conservation of mass,

$$VA = V_2 A_2 \quad \rightarrow \quad V_2 = V\frac{A}{A_2} = V\left(\frac{D}{D_2}\right)^2 \qquad (4)$$

Substitution of Eq. 4 into Eq. 3 yields

$$\left(z_1 - z_2\right) = \frac{V^2}{2g}\left(f\frac{L}{D} + \sum_{\text{pipe}} K_L + \left(\frac{D}{D_2}\right)^4 \left(\alpha_2 + K_{L,\text{contraction}}\right)\right) \qquad (5)$$

Equation 5 is an implicit equation for $V$ since the Darcy friction factor is a function of Reynolds number Re = $\rho VD/\mu$, as obtained from either the Moody chart or the Colebrook equation. The solution can be obtained by an iterative method, or through use of a mathematical equation solver like EES. The result is $V$ = 1.911 m/s, or to three significant digits, $V$ = 1.91 m/s, from which the volume flow rate is

$$\dot{V} = V\frac{\pi D^2}{4} = \left(1.911 \text{ m/s}\right)\frac{\pi\left(0.020 \text{ m}\right)^2}{4} = 6.01\times 10^{-4} \text{ m}^3\text{/s} \qquad (6)$$

In more common units, $\dot{V}$ = **36.0 Lpm** (liters per minute). The Reynolds number is $3.81 \times 10^4$.

***Discussion*** Since there is no net head across the pump at free delivery conditions, the pump could be removed (inlet and outlet pipes connected together without the pump), and the flow rate would be the same. Another way to think about this is that the pump's efficiency is zero at the free delivery operating point, so it is doing no useful work.

**14-25**

**Solution** We are to calculate the volume flow rate through a piping system in which the pipe is rough.

***Assumptions*** **1** Since the reservoir is large, the flow is nearly steady. **2** The water is incompressible. **3** The water is at room temperature. **4** The flow in the pipe is fully developed and turbulent, with $\alpha = 1.05$.

***Properties*** The density and viscosity of water at $T = 20^{\circ}$C are 998.0 kg/m$^3$ and $1.002 \times 10^{-3}$ kg/m·s respectively.

***Analysis*** The relative pipe roughness is $\varepsilon/D$ = (0.050 cm)/(2.0 cm) = 0.025 (very rough, as seen on the Moody chart). The calculations are identical to that of the previous problem, except for the pipe roughness. The result is $V$ = 1.597 m/s, or to three significant digits, $V$ = 1.60 m/s, from which the volume flow rate is $5.02 \times 10^{-4}$ m$^3$/s, or $\dot{V}$ = **30.1 Lpm**. The Reynolds number is $3.18 \times 10^4$. The volume flow rate is lower by about 16%. This agrees with our intuition, since pipe roughness leads to more pressure drop at a given flow rate.

***Discussion*** If the calculations of the previous problem are done on a computer, it is trivial to change $\varepsilon$ for the present calculations.

---

**14-26** 

**Solution** For a given pump and piping system, we are to calculate the volume flow rate and compare with that calculated for Problem 14-24.

***Assumptions*** **1** Since the reservoir is large, the flow is nearly steady. **2** The water is incompressible. **3** The water is at room temperature. **4** The flow in the pipe is fully developed and turbulent, with $\alpha = 1.05$.

***Properties*** The density and viscosity of water at $T = 20^{\circ}$C are 998.0 kg/m$^3$ and $1.002 \times 10^{-3}$ kg/m·s respectively.

***Analysis*** The calculations are identical to those of Problem 14-24 except that the pump's net head is not zero as in Problem 14-24, but instead is given in the problem statement. At the operating point, we match $H_{\text{available}}$ to $H_{\text{required}}$, yielding

$$H_{\text{available}} = H_{\text{required}} \quad \rightarrow \quad H_0 - a\dot{V}^2 = \frac{V^2}{2g}\left( f\frac{L}{D} + \sum_{\text{pipe}} K_L + \left(\frac{D}{D_2}\right)^4 \left(\alpha_2 + K_{L,\text{contraction}}\right)\right) - \left(z_1 - z_2\right) \qquad (1)$$

We re-write the second term on the left side of Eq. 1 in terms of average pipe velocity $V$ instead of volume flow rate, since $\dot{V} = V\pi D^2/4$, and solve for $V$,

$$V = \sqrt{\frac{H_0 + \left(z_1 - z_2\right)}{\frac{1}{2g}\left( f\frac{L}{D} + \sum_{\text{pipe}} K_L + \left(\frac{D}{D_2}\right)^4 \left(\alpha_2 + K_{L,\text{contraction}}\right)\right) + a\frac{\pi^2 D^4}{16}}} \qquad (2)$$

Equation 2 is an implicit equation for $V$ since the Darcy friction factor is a function of Reynolds number Re = $\rho VD/\mu$, as obtained from either the Moody chart or the Colebrook equation. The solution can be obtained by an iterative method, or through use of a mathematical equation solver like EES. The result is $V$ = 2.846 m/s, from which the volume flow rate is

$$\dot{V} = V\frac{\pi D^2}{4} = \left(2.846 \text{ m/s}\right)\frac{\pi\left(0.020 \text{ m}\right)^2}{4} = 8.942 \times 10^{-4} \frac{\text{m}^3}{\text{s}} \qquad (3)$$

In more common units, $\dot{V}$ = **53.6 Lpm**. This represents an increase of about 49% compared to the flow rate of Problem 14-24. This agrees with our expectations – adding a pump in the line produces a higher flow rate.

***Discussion*** Although there was a pump in Problem 14-24 as well, it was operating at free delivery conditions, implying that it was not contributing anything to the flow – that pump could be removed from the system with no change in flow rate. Here, however, the net head across the pump is about 5.34 m, implying that it is contributing useful head to the flow (in addition to the gravity head already present).

---

## 14-27E

**Solution** We are to calculate pump efficiency and estimate the BEP conditions.

*Properties* The density of water at 77°F is 62.24 lbm/ft$^3$.

*Analysis* (*a*) Pump efficiency is

*Pump efficiency*: $$\eta_{pump} = \frac{\rho g \dot{V} H}{bhp} \qquad (1)$$

We show the second row of data (at $\dot{V}$ = 4.0 gpm) as an example – the rest are calculated in a spreadsheet for convenience,

Pump performance data for water at 77°F.

| $\dot{V}$ (gpm) | *H* (ft) | *bhp* (hp) | $\eta_{pump}$ (%) |
|---|---|---|---|
| 0.0 | 19.0 | 0.06 | **0.0** |
| 4.0 | 18.5 | 0.064 | **29.2** |
| 8.0 | 17.0 | 0.069 | **49.7** |
| 12.0 | 14.5 | 0.074 | **59.3** |
| 16.0 | 10.5 | 0.079 | **53.6** |
| 20.0 | 6.0 | 0.08 | **37.8** |
| 24.0 | 0 | 0.078 | **0.0** |

$$\eta_{pump} = \frac{\left(62.24\text{ lbm/ft}^3\right)\left(32.2\frac{\text{ft}}{\text{s}^2}\right)\left(4.0\frac{\text{gal}}{\text{min}}\right)\left(18.5\text{ ft}\right)}{0.064\text{ hp}}\left(\frac{0.1337\text{ ft}^3}{\text{gal}}\right)\left(\frac{1\text{ min}}{60\text{ s}}\right)\times\left(\frac{\text{lbf}\cdot\text{s}^2}{32.2\text{ lbm}\cdot\text{ft}}\right)\left(\frac{\text{hp}\cdot\text{s}}{550\text{ ft}\cdot\text{lbf}}\right) = 0.292$$

or 29.2%. The results for all rows are shown in the table.

(*b*) The best efficiency point (BEP) occurs at approximately the fourth row of data: $\dot{V}^*$ = **12.0 gpm, *H** = 14.5 ft of head, *bhp** = 0.074 hp, and $\eta_{pump}$* = 59.3%**.

*Discussion* A more precise BEP could be obtained by curve-fitting the data, as in Problem 14-29.

## 14-28

**Solution** We are to convert the pump performance data to metric units and calculate pump efficiency.

*Properties* The density of water at $T$ = 20°C is 998.0 kg/m$^3$.

*Analysis* The conversions are straightforward, and the results are shown in the table. A sample calculation of the pump efficiency for the second row of data is shown below:

Pump performance data for water at 77°F.

| $\dot{V}$ (Lpm) | *H* (m) | *bhp* (W) | $\eta_{pump}$ (%) |
|---|---|---|---|
| 0.0 | 5.79 | 44.7 | **0.0** |
| 15.1 | 5.64 | 47.7 | **29.2** |
| 30.3 | 5.18 | 51.5 | **49.7** |
| 45.4 | 4.42 | 55.2 | **59.3** |
| 60.6 | 3.20 | 58.9 | **53.6** |
| 75.7 | 1.83 | 59.7 | **37.8** |
| 90.9 | 0.00 | 58.2 | **0.0** |

$$\eta_{pump} = \frac{\left(998.0\text{ kg/m}^3\right)\left(9.81\text{ m/s}^2\right)\left(15.1\text{ L/min}\right)\left(5.64\text{ m}\right)}{47.7\text{ W}}\left(\frac{1\text{ m}^3}{1000\text{ L}}\right)\left(\frac{1\text{ min}}{60\text{ s}}\right)\times\left(\frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}}\right)\left(\frac{\text{W}\cdot\text{s}}{\text{N}\cdot\text{m}}\right) = 0.292 = 29.2\%$$

The pump efficiency data are identical to those of the previous problem, as they must be, regardless of the system of units.

*Discussion* If the calculations of the previous problem are done on a computer, it is trivial to convert to metric units in the present calculations.

**14-29E** [*Also solved using EES on enclosed DVD*]

**Solution** We are to generate least-squares polynomial curve fits of a pump's performance curves, plot the curves, and calculate the BEP.

***Properties*** The density of water at 77°F is 62.24 lbm/ft³.

***Analysis*** The efficiencies for each data point in Table P14-27 are calculated in Problem 14-27. We use regression analysis to generate the least-squares fits. The equation and coefficients for *H* are

$$H = H_0 - a\dot{V}^2 \qquad H_0 = 19.0774 \text{ ft} \qquad a = 0.032996 \text{ ft/gpm}^2$$

Or, to 3 digits of precision, $H_0 = \mathbf{19.1\ ft}$ $a = \mathbf{0.0330\ ft/gpm^2}$

The equation and coefficients for *bhp* are

$$bhp = bhp_0 + a_1\dot{V} + a_2\dot{V}^2 \qquad bhp_0 = 0.0587 \text{ hp}$$

$$a_1 = \mathbf{0.00175\ hp/gpm} \qquad a_2 = \mathbf{-3.72\times10^{-5}\ hp/gpm^2}$$

The equation and coefficients for $\eta_{pump}$ are

$$\eta_{pump} = \eta_{pump,0} + a_1\dot{V} + a_2\dot{V}^2 + a_3\dot{V}^3 \qquad \eta_{pump,0} = 0.0523\%$$

$$a_1 = \mathbf{8.21\ \%/gpm} \qquad a_2 = \mathbf{-0.210\ \%/gpm^2} \qquad a_3 = \mathbf{-0.00546\ \%/gpm^3}$$

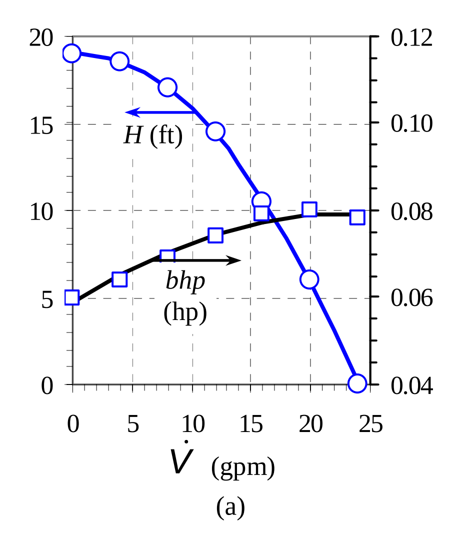
The tabulated data are plotted in Fig. 1 as symbols only. The fitted data are plotted on the same plots as lines only. The agreement is excellent.

The best efficiency point is obtained by differentiating the curve-fit expression for $\eta_{pump}$ with respect to volume flow rate, and setting the derivative to zero (solving the resulting quadratic equation for $\dot{V}^*$),

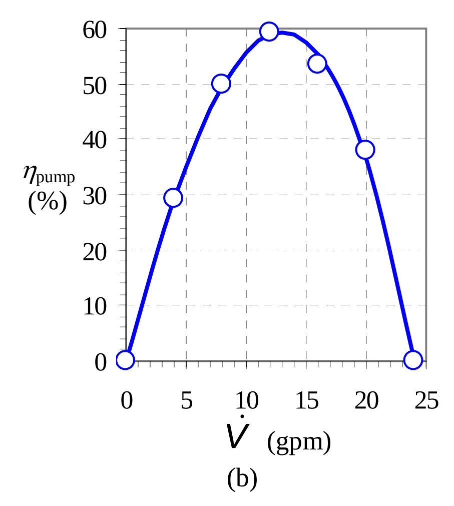
$$\frac{d\eta_{pump}}{d\dot{V}} = a_1 + 2a_2\dot{V} + 3a_3\dot{V}^2 = 0 \qquad \rightarrow \qquad \dot{V}^* = 12.966 \text{ GPM} \approx 13.0 \text{ gpm}$$

At this volume flow rate, the curve-fitted expressions for *H*, *bhp*, and $\eta_{pump}$ yield the operating conditions at the best efficiency point (to three digits each):

$$\dot{V}^* = \mathbf{13.0\ gpm}, \quad H^* = \mathbf{13.5\ ft}, \quad bhp^* = \mathbf{0.0752\ hp}, \quad \eta^* = \mathbf{59.2\%}$$




(a)




(b)

**FIGURE 1**
Pump performance curves: (a) *H* and *bhp* versus $\dot{V}$, and (b) $\eta_{pump}$ versus $\dot{V}$.

***Discussion*** This BEP is more precise than that of Problem 14-27 because of the curve fit. The other root of the quadratic is negative – obviously not the correct choice.

**14-30E**

**Solution** For a given pump and system requirement, we are to estimate the operating point.

***Assumptions*** **1** The flow is steady. **2** The water is at 77°F and is incompressible.

***Analysis*** The operating point is the volume flow rate at which $H_{required} = H_{available}$. We set the given expression for $H_{required}$ to the curve fit expression of Problem 14-29, $H_{available} = H_0 - a\dot{V}^2$, and obtain

*Operating point*:
$$\dot{V} = \sqrt{\frac{H_0 - (z_2 - z_1)}{a+b}} = \sqrt{\frac{19.0774 \text{ ft} - 15.5 \text{ ft}}{(0.032996 + 0.00986) \text{ ft/gpm}^2}} = \mathbf{9.14 \text{ gpm}}$$

At this volume flow rate, the net head of the pump is **16.3 ft**.

***Discussion*** At this operating point, the flow rate is lower than that at the BEP.

---

**14-31**

**Solution** We are to calculate pump efficiency and estimate the BEP conditions.

***Properties*** The density of water at 20°C is 998.0 kg/m³.

***Analysis*** (*a*) Pump efficiency is

*Pump efficiency*:
$$\eta_{pump} = \frac{\rho g \dot{V} H}{bhp} \quad (1)$$

We show the second row of data (at $\dot{V}$ = 6.0 Lpm) as an example – the rest are calculated in a spreadsheet for convenience,

**TABLE 1**
Pump performance data for water at 20°C.

| $\dot{V}$ (Lpm) | ***H*** (m) | ***bhp*** (W) | $\eta_{pump}$ (%) |
|---|---|---|---|
| 0.0 | 47.5 | 133 | **0.0** |
| 6.0 | 46.2 | 142 | **31.9** |
| 12.0 | 42.5 | 153 | **54.4** |
| 18.0 | 36.2 | 164 | **64.8** |
| 24.0 | 26.2 | 172 | **59.7** |
| 30.0 | 15.0 | 174 | **42.2** |
| 36.0 | 0.0 | 174 | **0.0** |

$$\eta_{pump} = \frac{(998.0 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(6.0 \text{ L/min})(46.2 \text{ m})}{142 \text{ W}}\left(\frac{1 \text{ m}^3}{1000 \text{ L}}\right)\left(\frac{1 \text{ min}}{60 \text{ s}}\right)\times\left(\frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}}\right)\left(\frac{\text{W}\cdot\text{s}}{\text{N}\cdot\text{m}}\right) = 0.319 = 31.9\%$$

The results for all rows are shown in Table 1.

(*b*) The best efficiency point (BEP) occurs at approximately the fourth row of data: $\dot{V}^*$ = **18.0 Lpm**, $H^*$ = **36.2 m of head,** $bhp^*$ = **164. W**, and $\eta_{pump}^*$ **= 64.8%**.

***Discussion*** A more precise BEP could be obtained by curve-fitting the data, as in the next problem.

**14-32** 

**Solution** We are to generate least-squares polynomial curve fits of a pump's performance curves, plot the curves, and calculate the BEP.

***Properties*** The density of water at 20°C is 998.0 kg/m³.

***Analysis*** The efficiencies for each data point in Table P14-31 were calculated in the previous problem. We use Regression analysis to generate the least-squares fits. The equation and coefficients for $H$ are

$$H = H_0 - a\dot{V}^2 \qquad H_0 = 47.6643 \text{ m} \qquad a = 0.0366453 \text{ m/Lpm}^2$$

Or, to 3 significant digits, $H_0 = \mathbf{47.7\ m}$ $\quad a = \mathbf{0.0366\ m/Lpm^2}$

The equation and coefficients for *bhp* are

$$bhp = bhp_0 + a_1\dot{V} + a_2\dot{V}^2 \qquad bhp_0 = 131.\text{ W}$$

$$a_1 = \mathbf{2.37\ W/Lpm} \qquad a_2 = \mathbf{-0.0317\ W/Lpm^2}$$

The equation and coefficients for $\eta_{pump}$ are

$$\eta_{pump} = \eta_{pump,0} + a_1\dot{V} + a_2\dot{V}^2 + a_3\dot{V}^3 \qquad \eta_{pump,0} = 0.152\%$$

$$a_1 = \mathbf{5.87\ \%/Lpm} \quad a_2 = \mathbf{-0.0905\ \%/Lpm^2} \quad a_3 = \mathbf{-0.00201\%/Lpm^3}$$

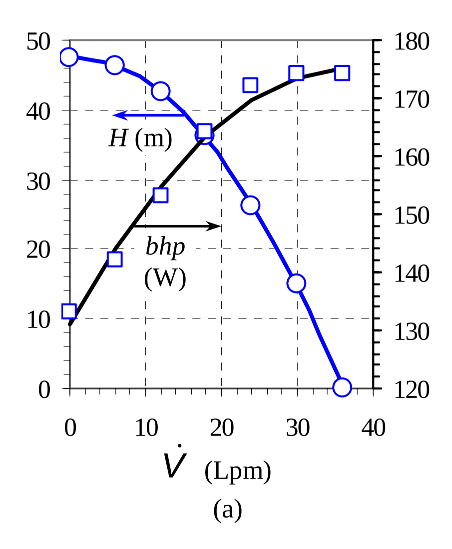
The tabulated data are plotted in Fig. 1 as symbols only. The fitted data are plotted on the same plots as lines only. The agreement is excellent.

The best efficiency point is obtained by differentiating the curve-fit expression for $\eta_{pump}$ with respect to volume flow rate, and setting the derivative to zero (solving the resulting quadratic equation for $\dot{V}*$),

$$\frac{d\eta_{pump}}{d\dot{V}} = a_1 + 2a_2\dot{V} + 3a_3\dot{V}^2 = 0 \qquad \rightarrow \qquad \dot{V}* = 19.6 \text{ Lpm}$$

At this volume flow rate, the curve-fitted expressions for $H$, *bhp*, and $\eta_{pump}$ yield the operating conditions at the best efficiency point (to three digits each):

$$\dot{V}* = \mathbf{19.6\ Lpm}, \quad H* = \mathbf{33.6\ m}, \quad bhp* = \mathbf{165\ W}, \quad \eta* = \mathbf{65.3\%}$$



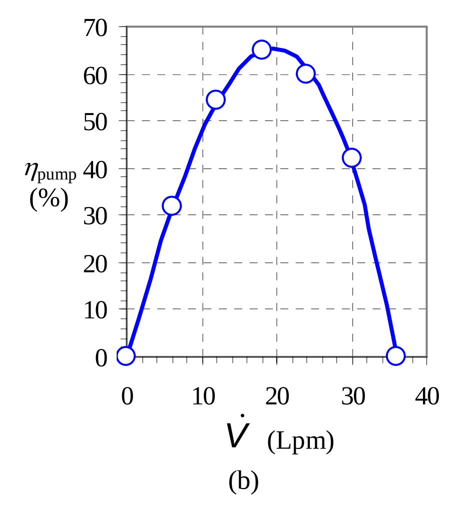




**FIGURE 1**
Pump performance curves: (a) $H$ and *bhp* versus $\dot{V}$, and (b) $\eta_{pump}$ versus $\dot{V}$.

***Discussion*** This BEP is more precise than that of the previous problem because of the curve fit. The other root of the quadratic is negative – obviously not the correct choice.

## 14-33

**Solution** For a given pump and system requirement, we are to estimate the operating point.

***Assumptions*** **1** The flow is steady. **2** The water is at 20°C and is incompressible.

***Analysis*** The operating point is the volume flow rate at which $H_{required} = H_{available}$. We set the given expression for $H_{required}$ to the curve fit expression of the previous problem, $H_{available} = H_0 - a\dot{V}^2$, and obtain

*Operating point*:
$$\dot{V} = \sqrt{\frac{H_0 - (z_2 - z_1)}{a + b}} = \sqrt{\frac{47.6643 \text{ m} - 10.0 \text{ m}}{(0.0366453 + 0.0185) \text{ m/Lpm}^2}} = \mathbf{26.1 \text{ Lpm}}$$

***Discussion*** At this operating point, the flow rate is higher than that at the BEP.

## 14-34



**Solution** We are to perform a regression analysis to estimate the shutoff head and free delivery of a pump, and then we are to determine if this pump is adequate for the system requirements.

***Assumptions*** **1** The water is incompressible. **2** The water is at room temperature.

***Properties*** The density of water at $T$ = 20°C is 998.0 kg/m³.

***Analysis*** (*a*) We perform a regression analysis, and obtain $\boldsymbol{H_0}$ **= 23.9 m** and $\boldsymbol{a}$ **= 0.00642 m/Lpm²**. The curve fit is reasonable, as seen in Fig. 1. **The shutoff head is estimated as 23.9 m of water column**. At the pump's free delivery, the net head is zero. Setting $H_{available}$ to zero in Eq. 1 gives

*Free delivery*:
$$\dot{V}_{max}^{\ \ 2} = \frac{H_0}{a} \quad \rightarrow \quad \dot{V}_{max} = \sqrt{\frac{H_0}{a}} = \sqrt{\frac{23.9 \text{ m}}{0.00642 \text{ m/(Lpm)}^2}} = 61.0 \text{ Lpm}$$

The free delivery is estimated as **61.0 Lpm**.

(*b*) At the required operating conditions, $\dot{V}$ = 57.0 Lpm, and the net head is converted to meters of water column for analysis,

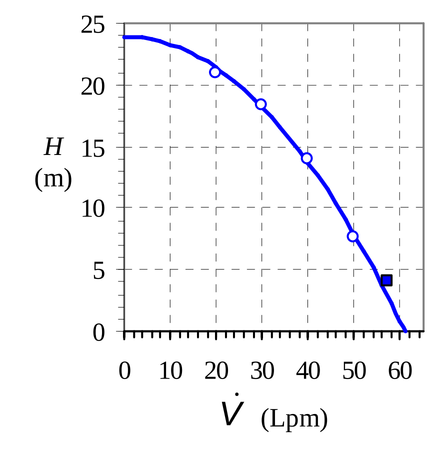



**FIGURE 1**
Tabulated data (circles) and curve-fitted data (line) for $H_{available}$ versus $\dot{V}$ for the given pump. The filled, square data point is the required operating point.

*Required operating head*:
$$H_{required} = \frac{(\Delta P)_{required}}{\rho g} = \frac{5.8 \text{ psi}}{(998. \text{ kg/m}^3)(9.81 \text{ m/s}^2)}\left(\frac{6{,}894.8 \text{ N/m}^2}{\text{psi}}\right)\left(\frac{\text{kg}\cdot\text{m}}{\text{s}^2\cdot\text{N}}\right) = 4.08 \text{ m}$$

As seen in Fig. 1, this operating point lies *above* the pump performance curve. Thus, **this pump is not quite adequate for the job at hand**.

***Discussion*** The operating point is also very close to the pump's free delivery, and therefore the pump efficiency would be low even if it *could* put out the required head.

**14-35E**

**Solution** We are to find the units of coefficient *a*, write $\dot{V}_{max}$ in terms of $H_0$ and *a*, and calculate the operating point of the pump.

***Assumptions*** **1** The flow is steady. **2** The water is incompressible.

***Analysis*** (*a*) Solving the given expression for *a* gives

*Coefficient a*:
$$a = \frac{H_0 - H_{available}}{\dot{V}^2} \quad \rightarrow \quad \boxed{\text{units of } a = \frac{\text{ft}}{\text{gpm}^2}} \qquad (1)$$

(*b*) At the pump's free delivery, the net head is zero. Setting $H_{available}$ to zero in the given expression gives

*Free delivery*:
$$\dot{V}_{max}{}^2 = \frac{H_0}{a} \quad \rightarrow \quad \boxed{\dot{V}_{max} = \sqrt{\frac{H_0}{a}}} \qquad (2)$$

(*c*) The operating point is obtained by matching the pump's performance curve to the system curve. Equating these gives

$$H_{available} = H_0 - a\dot{V}^2 = H_{required} = (z_2 - z_1) + b\dot{V}^2 \qquad (3)$$

After some algebra, Eq. 3 reduces to

*Operating point capacity*:
$$\boxed{\dot{V}_{operating} = \sqrt{\frac{H_0 - (z_2 - z_1)}{a+b}}} \qquad (4)$$

and the net pump head at the operating point is obtained by plugging Eq. 4 into the given expression,

*Operating point pump head*:
$$\boxed{H_{operating} = \frac{H_0 b + a(z_2 - z_1)}{a+b}} \qquad (5)$$

***Discussion*** Equation 4 reveals that $H_0$ must be greater than elevation difference ($z_2 - z_1$) in order to have a valid operating point. This agrees with our intuition, since the pump must be able to overcome the gravitational head between the tanks.

**14-36**

**Solution** We are to calculate the operating point of a given pipe/pump system.

***Assumptions*** **1** The water is incompressible. **2** The flow is steady since the reservoirs are large. **3** The water is at room temperature.

***Properties*** The density and viscosity of water at $T$ = 20°C are 998.0 kg/m$^3$ and 1.002 × 10$^{-3}$ kg/m·s respectively.

***Analysis*** The operating point is obtained by matching the pump's performance curve to the system curve,

*Operating point:*
$$H_{\text{available}} = H_0 - a\dot{V}^2 = H_{\text{required}} = (z_2 - z_1) + b\dot{V}^2$$

from which we solve for the volume flow rate (capacity) at the operating point,

$$\dot{V}_{\text{operating}} = \sqrt{\frac{H_0 - (z_2 - z_1)}{a+b}} = \sqrt{\frac{5.30\text{ m} - 3.52\text{ m}}{(0.0453 + 0.0261)\text{ m/Lpm}^2}} = \mathbf{4.99\ Lpm}$$

and for the net pump head at the operating point,

$$H_{\text{operating}} = \frac{H_0 b + a(z_2 - z_1)}{a+b} = \frac{(5.30\text{ m})(0.0261\text{ m}) + (0.0453\text{ m})(3.52\text{ m})}{(0.0453\text{ m}) + (0.0261\text{ m})} = \mathbf{4.17\ m}$$

***Discussion*** The water properties $\rho$ and $\mu$ are not needed because the system curve ($H_{\text{required}}$ versus $\dot{V}$) is provided here.

## 14-37E

**Solution** For a given pump and system, we are to calculate the capacity.

***Assumptions*** **1** The water is incompressible. **2** The flow is nearly steady since the reservoirs are large. **3** The water is at room temperature.

***Properties*** The kinematic viscosity of water at $T = 68^{o}$F is $1.055 \times 10^{-5}$ ft$^2$/s.

***Analysis*** We apply the energy equation in head form between the inlet reservoir's free surface (1) and the outlet reservoir's free surface (2),

$$H_{\text{required}} = h_{\text{pump,u}} = \frac{P_2 - P_1}{\rho g} + \frac{\alpha_2 V_2^2 - \alpha_1 V_1^2}{2g} + (z_2 - z_1) + h_{\text{turbine}} + h_{L,\text{total}} \qquad (1)$$

Since both free surfaces are at atmospheric pressure, $P_1 = P_2 = P_{\text{atm}}$, and the first term on the right side of Eq. 1 vanishes. Furthermore, since there is no flow, $V_1 = V_2 = 0$, and the second term also vanishes. There is no turbine in the control volume, so the second-to-last term is zero. Finally, the irreversible head losses are composed of both major and minor losses, but the pipe diameter is constant throughout. Equation 1 therefore reduces to

$$H_{\text{required}} = (z_2 - z_1) + h_{L,\text{total}} = (z_2 - z_1) + \left( f\frac{L}{D} + \sum K_L \right)\frac{V^2}{2g} \qquad (2)$$

The dimensionless roughness factor is $\varepsilon/D = 0.0011/1.20 = 9.17 \times 10^{-4}$, and the sum of all the minor loss coefficients is

$$\sum K_L = 0.5 + 2.0 + 6.8 + (3 \times 0.34) + 1.05 = 11.37$$

The pump/piping system operates at conditions where the available pump head equals the required system head. Thus, we equate the given expression and Eq. 2 to find the operating point,

$$H_{\text{available}} = H_{\text{required}} \quad \rightarrow \quad H_0 - a\frac{\pi^2 D^4}{16}V^2 = (z_2 - z_1) + \left( f\frac{L}{D} + \sum K_L \right)\frac{V^2}{2g} \qquad (3)$$

where we have written the volume flow rate in terms of average velocity through the pipe,

*Volume flow rate in terms of average velocity*: $$\dot{V} = V\frac{\pi D^2}{4} \qquad (4)$$

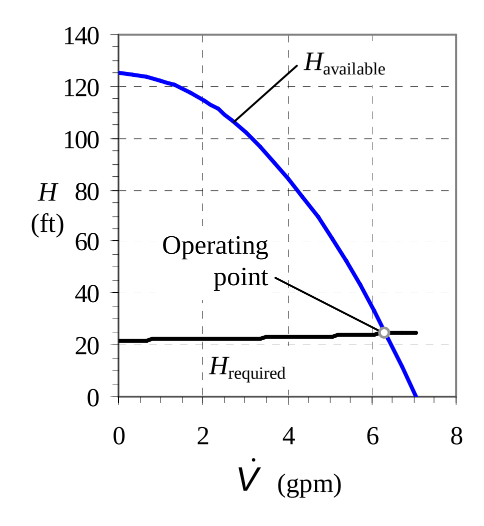
Equation 3 is an implicit equation for $V$ since the Darcy friction factor $f$ is a function of Reynolds number Re = $\rho VD/\mu$ = $VD/\nu$, as obtained from either the Moody chart or the Colebrook equation. The solution can be obtained by an iterative method, or through use of a mathematical equation solver like EES. The result is $V$ = 1.80 ft/s, from which the volume flow rate is $\dot{V}$ = **6.34 gpm**. The Reynolds number is $1.67 \times 10^4$.

***Discussion*** We verify our results by comparing $H_{\text{available}}$ (given) and $H_{\text{required}}$ (Eq. 2) at this flow rate: $H_{\text{available}}$ = 24.4 ft and $H_{\text{required}}$ = 24.4 ft.

## 14-38E

**Solution** We are to plot $H_{required}$ and $H_{available}$ versus $\dot{V}$, and indicate the operating point.

***Analysis*** We use the equations of the previous problem, with the same constants and parameters, to generate the plot shown. The operating point is the location where the two curves intersect. The values of $H$ and $\dot{V}$ at the operating point match those of the previous problem, as they should.




***Discussion*** A plot like this, in fact, is an alternate method of obtaining the operating point. In this case, the curve of $H_{required}$ is fairly flat, indicating that the majority of the required pump head is attributed to elevation change, while a small fraction is attributed to major and minor head losses through the piping system.

---

## 14-39E

**Solution** We are to re-calculate volume flow rate for a piping system with a much longer pipe, and we are to compare with the previous results.

***Analysis*** All assumptions, properties, dimensions, and parameters are identical to those of the previous problem, except that total pipe length $L$ is longer. We repeat the calculations and find that $V$ = 1.68 ft/s, from which the volume flow rate is $\dot{V}$ = **5.93 gpm**, and the net head of the pump is 37.0 ft. The Reynolds number for the flow in the pipe is $1.56 \times 10^4$. **The volume flow rate has decreased by about 6.5%**.

***Discussion*** The decrease in volume flow rate is smaller than we may have suspected. This is because the majority of the pump work goes into raising the elevation of the water. In addition, as seen in the plot from the previous problem, the pump performance curve is quite steep near these flow rates – a significant change in required net head leads to a much less significant change in volume flow rate.

---

## 14-40E



**Solution** We are to perform a regression analysis to translate tabulated pump performance data into an analytical expression, and then use this expression to predict the volume flow rate through a piping system.

***Assumptions*** **1** The water is incompressible. **2** The flow is nearly steady since the reservoirs are large. **3** The water is at room temperature.

***Properties*** For water at $T = 68^{\circ}$F, $\mu = 6.572 \times 10^{-4}$ lbm/ft·s, and $\rho = 62.31$ lbm/ft$^3$, from which $\nu = 1.055 \times 10^{-5}$ ft$^2$/s.

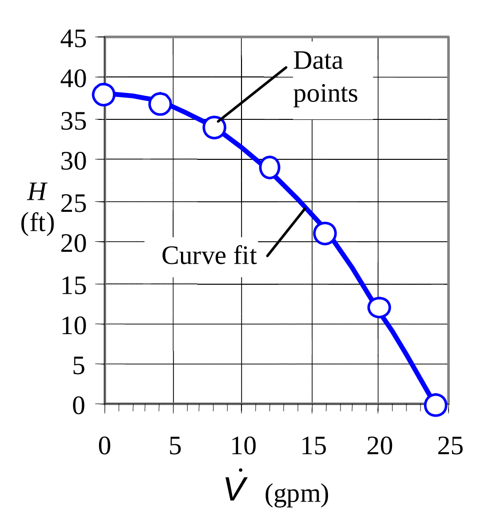



**FIGURE 1**
Tabulated data (symbols) and curve-fitted data (line) for $H_{\text{available}}$ versus $\dot{V}$ for the proposed pump.

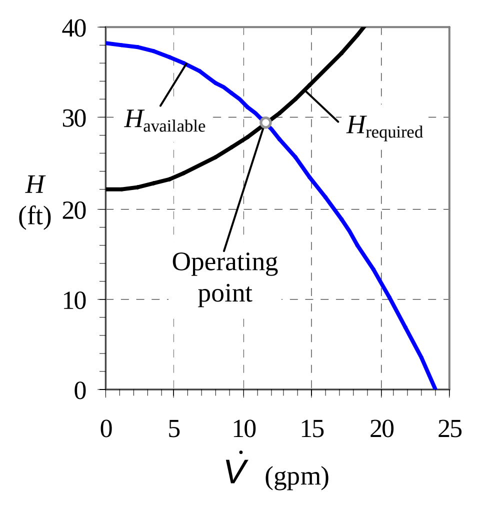



**FIGURE 2**
$H_{\text{available}}$ and $H_{\text{required}}$ versus $\dot{V}$ for a piping system with pump; the operating point is also indicated, where the two curves meet.

***Analysis*** (*a*) We perform a regression analysis, and obtain $\boldsymbol{H_0}$ **= 38.15 ft** and $\boldsymbol{a}$ **= 0.06599 ft/gpm$^2$**. The curve fit is very good, as seen in Fig. 1.

(*b*) We repeat the calculations of Problem 14-37 with the new pump performance coefficients, and find that $V = 3.29$ ft/s, from which the volume flow rate is $\dot{V}$ = **11.6 gpm**, and the net head of the pump is 29.3 ft. The Reynolds number for the flow in the pipe is $3.05 \times 10^4$. **The volume flow rate has increased by about 83%**. **Paul is correct** – this pump performs much better, nearly doubling the flow rate.

(*c*) A plot of net head versus volume flow rate is shown in Fig. 2.

***Discussion*** This pump is more appropriate for the piping system at hand.

**14-41**

**Solution** For a given pump and system, we are to calculate the capacity.

***Assumptions*** **1** The water is incompressible. **2** The flow is nearly steady since the reservoirs are large. **3** The water is at room temperature.

***Properties*** The density and viscosity of water at $T = 20^{o}$C are 998.0 kg/m$^3$ and $1.002 \times 10^{-3}$ kg/m·s respectively.

***Analysis*** We apply the energy equation in head form between the inlet reservoir's free surface (1) and the outlet reservoir's free surface (2),

$$H_{\text{required}} = h_{\text{pump,u}} = \frac{P_2 - P_1}{\rho g} + \frac{\alpha_2 V_2^2 - \alpha_1 V_1^2}{2g} + (z_2 - z_1) + h_{\text{turbine}} + h_{L,\text{total}} \qquad (1)$$

Since both free surfaces are at atmospheric pressure, $P_1 = P_2 = P_{\text{atm}}$, and the first term on the right side of Eq. 1 vanishes. Furthermore, since there is no flow, $V_1 = V_2 = 0$, and the second term also vanishes. There is no turbine in the control volume, so the second-to-last term is zero. Finally, the irreversible head losses are composed of both major and minor losses, but the pipe diameter is constant throughout. Equation 1 therefore reduces to

$$H_{\text{required}} = (z_2 - z_1) + h_{L,\text{total}} = (z_2 - z_1) + \left( f\frac{L}{D} + \sum K_L \right)\frac{V^2}{2g} \qquad (2)$$

The dimensionless roughness factor is

$$\frac{\varepsilon}{D} = \frac{0.25\text{ mm}}{2.03\text{ cm}}\left(\frac{1\text{ cm}}{10\text{ mm}}\right) = 0.0123$$

The sum of all the minor loss coefficients is

$$\sum K_L = 0.5 + 17.5 + (5 \times 0.92) + 1.05 = 23.65$$

The pump/piping system operates at conditions where the available pump head equals the required system head. Thus, we equate the given expression and Eq. 2 to find the operating point,

$$H_{\text{available}} = H_{\text{required}} \quad \rightarrow \quad H_0 - a\frac{\pi^2 D^4}{16}V^2 = (z_2 - z_1) + \left( f\frac{L}{D} + \sum K_L \right)\frac{V^2}{2g} \qquad (3)$$

where we have written the volume flow rate in terms of average velocity through the pipe,

$$\dot{V} = V\frac{\pi D^2}{4}$$

Equation 3 is an implicit equation for $V$ since the Darcy friction factor $f$ is a function of Reynolds number Re = $\rho VD/\mu$, as obtained from either the Moody chart or the Colebrook equation. The solution can be obtained by an iterative method, or through use of a mathematical equation solver like EES. The result is $V$ = 0.59603 ≈ 0.596 m/s, from which the volume flow rate is $\dot{V}$ = **11.6 Lpm**. The Reynolds number is $1.21 \times 10^4$.

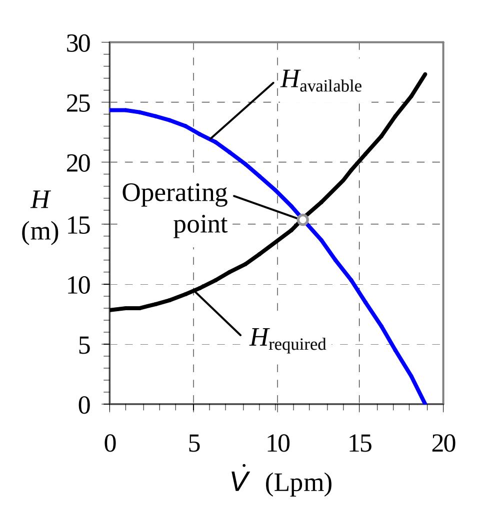
***Discussion*** We verify our results by comparing $H_{\text{available}}$ (given) and $H_{\text{required}}$ (Eq. 2) at this flow rate: $H_{\text{available}}$ = 15.3 m and $H_{\text{required}}$ = 15.3 m.

**14-42**

**Solution** We are to plot $H_{\text{required}}$ and $H_{\text{available}}$ versus $\dot{V}$, and indicate the operating point.

***Analysis*** We use the equations of the previous problem, with the same constants and parameters, to generate the plot shown. The operating point is the location where the two curves intersect. The values of $H$ and $\dot{V}$ at the operating point match those of the previous problem, as they should.

***Discussion*** A plot like this, in fact, is an alternate method of obtaining the operating point.




---

**14-43**

**Solution** We are to re-calculate volume flow rate for a piping system with a smaller elevation difference, and we are to compare with the previous results.

***Analysis*** All assumptions, properties, dimensions, and parameters are identical to those of the previous problem, except that the elevation difference between reservoir surfaces ($z_2 - z_1$) is smaller. We repeat the calculations and find that $V$ = 0.682 m/s, from which the volume flow rate is $\dot{V}$ = **13.2 Lpm**, and the net head of the pump is 12.5 m. The Reynolds number for the flow in the pipe is $1.38 \times 10^4$. **The volume flow rate has increased by about 14%**.

***Discussion*** The increase in volume flow rate is modest. This is because only about half of the pump work goes into raising the elevation of the water – the other half goes into overcoming irreversible losses.

---

## 14-44 

**Solution** We are to perform a regression analysis to translate tabulated pump performance data into an analytical expression, and then use this expression to predict the volume flow rate through a piping system.

***Assumptions*** **1** The water is incompressible. **2** The flow is nearly steady since the reservoirs are large. **3** The water is at room temperature.

***Properties*** The density and viscosity of water at $T$ = 20°C are 998.0 kg/m$^3$ and 1.002 × 10$^{-3}$ kg/m·s respectively.

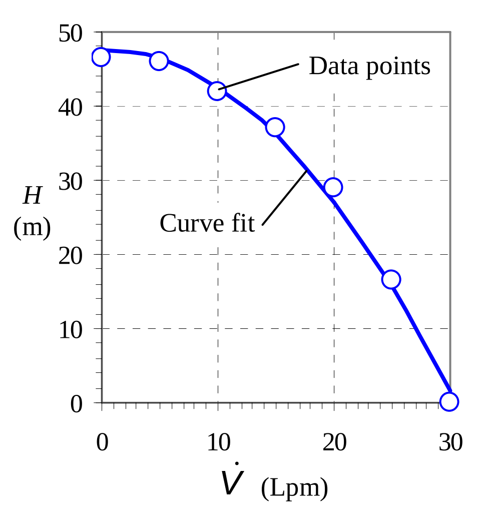



**FIGURE 1**
Tabulated data (symbols) and curve-fitted data (line) for $H_{\text{available}}$ versus $\dot{V}$ for the proposed pump.

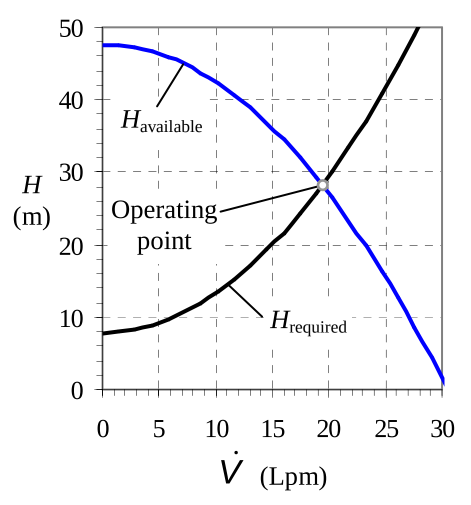



**FIGURE 2**
$H_{\text{available}}$ and $H_{\text{required}}$ versus $\dot{V}$ for a piping system with pump; the operating point is also indicated, where the two curves meet.

***Analysis*** (*a*) We perform a regression analysis, and obtain $\boldsymbol{H_0}$ **= 47.6 m** and $\boldsymbol{a}$ **= 0.05119 m/Lpm$^2$**. The curve fit is reasonable, as seen in Fig. 1.

(*b*) We repeat the calculations of Problem 14-41 with the new pump performance coefficients, and find that $V$ = 1.00 m/s, from which the volume flow rate is $\dot{V}$ = **19.5 Lpm**, and the net head of the pump is 28.3 m. The Reynolds number for the flow in the pipe is 2.03 × 10$^4$. **The volume flow rate has increased by about 69%**. **April's goal has *not* been reached**. She will need to search for an even stronger pump.

(*c*) A plot of net head versus volume flow rate is shown in Fig. 2.

***Discussion*** As is apparent from Fig. 2, the required net head increases rapidly with increasing volume flow rate. Thus, doubling the flow rate would require a significantly heftier pump.

## 14-45

**Solution** We are to calculate the volume flow rate when the pipe diameter of a piping/pump system is doubled.

***Analysis*** The analysis is identical to that of Problem 14-41 except for the diameter change. The calculations yield $V$ = 0.19869 ≈ 0.199 m/s, from which the volume flow rate is $\dot{V}$ = **15.4 Lpm**, and the net head of the pump is 8.25 m. The Reynolds number for the flow in the pipe is 8.03 × $10^3$. **The volume flow rate has increased by about 33%**. This agrees with our intuition since irreversible head losses go down significantly by increasing pipe diameter.

***Discussion*** The gain in volume flow rate is significant because the irreversible head losses contribute to about half of the total pump head requirement in the original problem.

---

## 14-46

**Solution** We are to compare Reynolds numbers for a pipe flow system – the second case having a pipe diameter twice that of the first case.

***Properties*** The density and viscosity of water at $T$ = 20°C are 998.0 kg/m$^3$ and 1.002 × $10^{-3}$ kg/m·s respectively.

***Analysis*** From the results of the two problems, the Reynolds number of the first case is

*Case* 1 *(D* = 2.03 cm*)*:

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{\left(998 \text{ kg/m}^3\right)\left(0.59603 \text{ m/s}\right)\left(0.0203 \text{ m}\right)}{1.002\times10^{-3} \text{ kg/m}^3} = 1.21\times10^4$$

and that of the second case is

*Case* 2 *(D* = 4.06 cm*)*:

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{\left(998 \text{ kg/m}^3\right)\left(0.19869 \text{ m/s}\right)\left(0.0406 \text{ m}\right)}{1.002\times10^{-3} \text{ kg/m}^3} = 0.803\times10^4$$

Thus, **the Reynolds number of the larger diameter pipe is *smaller* than that of the smaller diameter pipe**. This may be somewhat surprising, but since average pipe velocity scales as the inverse of pipe diameter squared, Reynolds number increases linearly with pipe diameter due to the $D$ in the numerator, but decreases quadratically with pipe diameter due to the $V$ in the numerator. The net effect is a decrease in Re with pipe diameter when $\dot{V}$ is the same. In this problem, $\dot{V}$ increases somewhat as the diameter is doubled, but not enough to increase the Reynolds number.

***Discussion*** At first glance, most people would think that Reynolds number increases as both diameter and volume flow rate increase, but this is not always the case.

---

## 14-47

**Solution** We are to compare the volume flow rate in a piping system with and without accounting for minor losses.

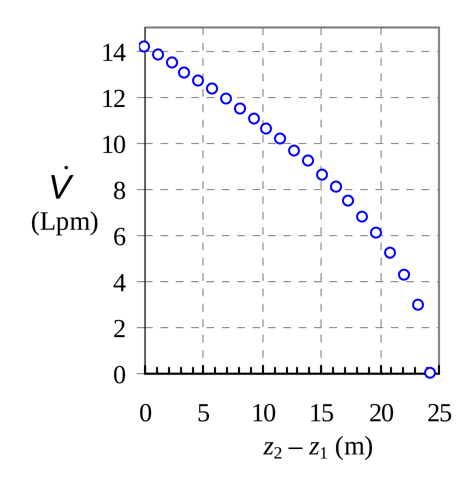
***Analysis*** The analysis is identical to that of Problem 14-41, except we ignore all the minor losses. The calculations yield $V$ = 0.604 m/s, from which the volume flow rate is $\dot{V}$ = **11.7 Lpm**, and the net head of the pump is 15.1 m. The Reynolds number for the flow in the pipe is 1.22 × $10^4$. **The volume flow rate has increased by about 1.3%**. Thus, **minor losses are nearly negligible in this calculation**. This agrees with our intuition since the pipe is very long.

***Discussion*** Since the Colebrook equation is accurate to at most 5%, a 1.3% change is well within the error. Nevertheless, it is not excessively difficult to include the minor losses, especially when solving the problem on a computer.

---

**14-48** 

**Solution** We are to examine how increasing ($z_2 - z_1$) affects the volume flow rate of water pumped by the water pump.




***Assumptions*** **1** The flow at any instant of time is still considered quasi-steady, since the surface level of the upper reservoir rises very slowly. **2** The minor losses, dimensions, etc., fluid properties, and all other assumptions are identical to those of Problem 14-41 except for the elevation difference ($z_2 - z_1$).

***Analysis*** We repeat the calculations of Problem 14-41 for several values of ($z_2 - z_1$), ranging from 0 to $H_0$, the shutoff head of the pump, since above the shutoff head, the pump cannot overcome the elevation difference. **The volume flow rate is zero at the shutoff head of the pump**. The data are plotted here. As expected, the volume flow rate decreases as ($z_2 - z_1$) increases, starting at a maximum flow rate of about 14.1 Lpm when there is no elevation difference, and reaching zero (no flow) when ($z_2 - z_1$) = $H_0$ = 24.4 m. **The curve is *not* linear**, since neither the Darcy friction factor nor the pump performance curve are linear. If ($z_2 - z_1$) were increased beyond $H_0$, the pump would not be able to handle the elevation difference. Despite its valiant efforts, with blades spinning as hard as they could, **the water would flow backwards through the pump**.

***Discussion*** You may wish to think of the backward-flow through the pump as a case in which the pump efficiency is *negative*. In fact, at ($z_2 - z_1$) = $H_0$, the pump could be replaced by a closed valve to keep the water from draining from the upper reservoir to the lower reservoir.

**14-49E**

**Solution** We are to estimate the operating point of a given fan and duct system.

***Assumptions*** **1** The flow is steady. **2** The concentration of contaminants in the air is low; the fluid properties are those of air alone. **3** The air is at standard temperature and pressure (STP), and is incompressible. **4** The air flowing in the duct is turbulent with $\alpha = 1.05$.

***Properties*** For air at STP ($T$ = 77°F, $P$ = 14.696 psi = 2116.2 lbf/ft$^2$), $\mu = 1.242 \times 10^{-5}$ lbm/ft·s, $\rho$ = 0.07392 lbm/ft$^3$, and $\nu = 1.681 \times 10^{-4}$ ft$^2$/s. The density of water at STP (for conversion to inches of water head) is 62.24 lbm/ft$^3$.

***Analysis*** We apply the steady energy equation along a streamline from point 1 in the stagnant air region in the room to point 2 at the duct outlet,

*Required net head*:
$$H_{\text{required}} = \frac{P_2 - P_1}{\rho g} + \frac{\alpha_2 V_2^2 - \alpha_1 V_1^2}{2g} + \left(z_2 - z_1\right) + h_{L,\text{total}} \qquad (1)$$

At point 1, $P_1$ is equal to $P_{\text{atm}}$, and at point 2, $P_2$ is also equal to $P_{\text{atm}}$ since the jet discharges into the outside air on the roof of the building. Thus the pressure terms cancel out in Eq. 1. We ignore the air speed at point 1 since it is chosen (wisely) far enough away from the hood inlet so that the air is nearly stagnant. Finally, the elevation difference is neglected for gases. Equation 1 reduces to

$$H_{\text{required}} = \frac{\alpha_2 V_2^2}{2g} + h_{L,\text{total}} \qquad (2)$$

The total head loss in Eq. 2 is a combination of major and minor losses, and depends on volume flow rate. Since the duct diameter is constant,

*Total irreversible head loss*:
$$h_{L,\text{total}} = \left(f\frac{L}{D} + \sum K_L\right)\frac{V^2}{2g} \qquad (3)$$

The required net head of the fan is thus

$$H_{\text{required}} = \left(\alpha_2 + f\frac{L}{D} + \sum K_L\right)\frac{V^2}{2g} \qquad (4)$$

To find the operating point, we equate $H_{\text{available}}$ and $H_{\text{required}}$, being careful to keep consistent units. Note that the required head in Eq. 4 is expressed naturally in units of equivalent column height of the pumped fluid, which is air in this case. However, the available net head (given) is in terms of equivalent water column height. We convert constants $H_0$ and $a$ to inches of *air* column for consistency by multiplying by the ratio of water density to air density,

$$H_{0,\text{ inch water}}\rho_{\text{water}} = H_{0,\text{ inch air}}\rho_{\text{air}} \quad \rightarrow \quad H_{0,\text{ inch air}} = H_{0,\text{ inch water}}\frac{\rho_{\text{water}}}{\rho_{\text{air}}}$$

and similarly,

$$a_{(\text{inch air})/\text{SCFM}^2} = a_{(\text{inch water})/\text{SCFM}^2}\frac{\rho_{\text{water}}}{\rho_{\text{air}}}$$

We re-write the given expression in terms of average duct velocity rather than volume flow rate,

*Available net head*:
$$H_{\text{available}} = H_0 - a\frac{\pi^2 D^4}{16}V^2 \qquad (5)$$

again taking care to keep consistent units. Equating Eqs. 4 and 5 yields

*Operating point*:
$$H_{\text{available}} = H_{\text{required}} \quad \rightarrow \quad H_0 - a\frac{\pi^2 D^4}{16}V^2 = \left(\alpha_2 + f\frac{L}{D} + \sum K_L\right)\frac{V^2}{2g} \qquad (6)$$

The dimensionless roughness factor is $\varepsilon/D = 0.0059/9.06 = 6.52 \times 10^{-4}$, and the sum of all the minor loss coefficients is

*Minor losses*: $\sum K_L = 4.6 + (3 \times 0.21) + 1.8 = 7.03$

Note that there is no minor loss associated with the exhaust, since point 2 is taken at the exit plane of the duct, and does not include irreversible losses associated with the turbulent jet. Equation 6 is an implicit equation for $V$ since the Darcy friction factor is a function of Reynolds number Re = $\rho VD/\mu = VD/\nu$, as obtained from either the Moody chart or the Colebrook equation. The solution can be obtained by an iterative method, or through use of a mathematical equation solver like EES. The result is $V$ = 16.8 ft/s, from which the volume flow rate is $\dot{V}$ = **452 SCFM**. The Reynolds number is $7.63 \times 10^4$.

***Discussion*** We verify our results by comparing $H_{available}$ (Eq. 1) and $H_{required}$ (Eq. 5) at this flow rate: $H_{available}$ = 0.566 inches of water and $H_{required}$ = 0.566 inches of water.

---

## 14-50E

**Solution** We are to calculate the value of $K_{L,\text{ damper}}$ such that the volume flow rate through the duct decreases by 50%.

***Analysis*** All assumptions and properties are the same as those of the previous problem. We set the volume flow rate to $\dot{V}$ = 226 SCFM, one-half of the previous result, and solve for $K_{L,\text{ damper}}$. The result is $\boldsymbol{K_{L,\text{ damper}} = 112}$, significantly higher than the value of 1.8 for the fully open case.

***Discussion*** Because of the nonlinearity of the problem, we cannot simply double the damper's loss coefficient in order to decrease the flow rate by a factor of two. Indeed, the minor loss coefficient must be increased by a factor of more than 60. If a computer was used for the calculations of the previous problem, the solution here is most easily obtained by trial and error.

---

## 14-51E

**Solution** We are to estimate the volume flow rate at the operating point without accounting for minor losses, and then we are to compare with the previous results.

***Analysis*** All assumptions and properties are the same as those of Problem 14-49, except that we ignore all minor losses (we set $\Sigma K_L = 0$). The resulting volume flow rate at the operating point is $\dot{V}$ = **503 SCFM**, approximately 11% higher than for the case with minor losses taken into account. In this problem, minor losses are indeed "minor", although they are not negligible. We should not be surprised at this result, since there are several minor losses, and the duct is not extremely long ($L/D$ is only 45.0).

***Discussion*** An error of 11% may be acceptable in this type of problem. However, since it is not difficult to account for minor losses, especially if the calculations are performed on a computer, it is wise not to ignore these terms.

---

**14-52**

**Solution** We are to estimate the operating point of a given fan and duct system.

***Assumptions*** **1** The flow is steady and incompressible. **2** The concentration of contaminants is low; the fluid properties are those of air alone. **3** The air is at 25°C and 101,300 Pa. **4** The air flowing in the duct is turbulent with $\alpha$ = 1.05.

***Properties*** For air at 25°C, $\mu = 1.849 \times 10^{-5}$ kg/m·s, $\rho = 1.184$ kg/m$^3$, and $\nu = 1.562 \times 10^{-5}$ m$^2$/s. The density of water at STP (for conversion to water head) is 997.0 kg/m$^3$.

***Analysis*** We apply the steady energy equation along a streamline from point 1 in the stagnant air region in the room to point 2 at the duct outlet,

$$H_{\text{required}} = \frac{P_2 - P_1}{\rho g} + \frac{\alpha_2 V_2^2 - \alpha_1 V_1^2}{2g} + (z_2 - z_1) + h_{L,\text{total}} \qquad (1)$$

$P_1$ is equal to $P_{\text{atm}}$, and $P_2$ is also equal to $P_{\text{atm}}$ since the jet discharges into outside air on the roof of the building. Thus the pressure terms cancel out in Eq. 1. We ignore the air speed at point 1 since it is chosen (wisely) far enough away from the hood inlet so that the air is nearly stagnant. Finally, the elevation difference is neglected for gases. Equation 1 reduces to

*Required net head*:
$$H_{\text{required}} = \frac{\alpha_2 V_2^2}{2g} + h_{L,\text{total}} \qquad (2)$$

The total head loss in Eq. 2 is a combination of major and minor losses. Since the duct diameter is constant,

*Total irreversible head loss*:
$$h_{L,\text{total}} = \left( f\frac{L}{D} + \sum K_L \right)\frac{V^2}{2g} \qquad (3)$$

The required net head of the fan is thus

$$H_{\text{required}} = \left( \alpha_2 + f\frac{L}{D} + \sum K_L \right)\frac{V^2}{2g} \qquad (4)$$

To find the operating point, we equate $H_{\text{available}}$ and $H_{\text{required}}$, being careful to keep consistent units. Note that the required head in Eq. 4 is expressed naturally in units of equivalent column height of the pumped fluid, which is air in this case. However, the available net head (given) is in terms of equivalent *water* column height. We convert constants $H_0$ and $a$ in Eq. 1 to mm of *air* column for consistency by multiplying by the ratio of water density to air density,

$$H_{0,\text{ mm water}}\rho_{\text{water}} = H_{0,\text{ mm air}}\rho_{\text{air}} \quad \rightarrow \quad H_{0,\text{ mm air}} = H_{0,\text{ mm water}}\frac{\rho_{\text{water}}}{\rho_{\text{air}}} \quad \text{and} \quad a_{(\text{mm air})/\text{LPM}^2} = a_{(\text{mm water})/\text{LPM}^2}\frac{\rho_{\text{water}}}{\rho_{\text{air}}}$$

We re-write the given expression in terms of average duct velocity rather than volume flow rate,

*Available net head*:
$$H_{\text{available}} = H_0 - a\frac{\pi^2 D^4}{16}V^2 \qquad (5)$$

Equating Eqs. 4 and 5 yields the operating point,

$$H_{\text{available}} = H_{\text{required}} \quad \rightarrow \quad H_0 - a\frac{\pi^2 D^4}{16}V^2 = \left( \alpha_2 + f\frac{L}{D} + \sum K_L \right)\frac{V^2}{2g} \qquad (6)$$

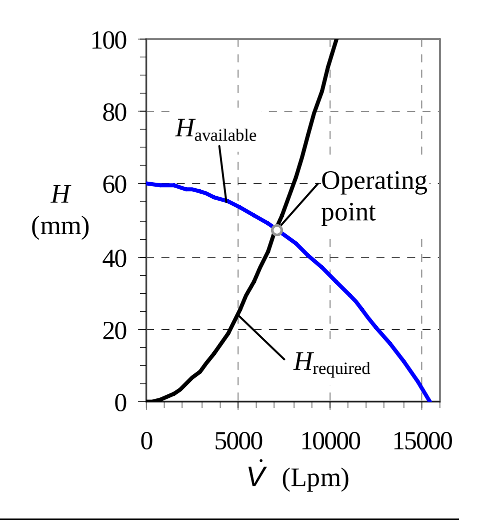
The dimensionless roughness factor is $\varepsilon/D$ = 0.15/150 = 1.00 × 10$^{-3}$, and the sum of all the minor loss coefficients is $\sum K_L = 3.3 + (3\times 0.21) + 1.8 + 0.36 + 6.6 = 12.69$. Note that there is no minor loss associated with the exhaust, since point 2 is at the exit plane of the duct, and does not include irreversible losses associated with the turbulent jet. Equation 6 is an implicit equation for $V$ since the Darcy friction factor is a function of Reynolds number Re = $\rho VD/\mu = VD/\nu$, as obtained from the Moody chart or the Colebrook equation. The solution can be obtained by an iterative method, or through use of a mathematical equation solver like EES. The result is $V$ = 6.71 m/s, from which the volume flow rate is $\dot{V}$ = **7090 Lpm**.

***Discussion*** We verify our results by comparing $H_{\text{available}}$ (given) and $H_{\text{required}}$ (Eq. 5) at this flow rate: $H_{\text{available}}$ = 47.4 mm of water and $H_{\text{required}}$ = 47.4 mm of water, both of which are equivalent to 40.0 m of air column.

## 14-53

**Solution** We are to plot $H_{\text{required}}$ and $H_{\text{available}}$ versus $\dot{V}$, and indicate the operating point.

***Analysis*** We use the equations of the previous problem, with the same constants and parameters, to generate the plot shown. The operating point is the location where the two curves intersect. The values of $H$ and $\dot{V}$ at the operating point match those of the previous problem, as they should.




***Discussion*** A plot like this, in fact, is an alternate method of obtaining the operating point. The operating point is at a volume flow rate near the center of the plot, indicating that the fan efficiency is probably reasonably high.

---

## 14-54

**Solution** We are to estimate the volume flow rate at the operating point without accounting for minor losses, and then we are to compare with the previous results.

***Analysis*** All assumptions and properties are the same as those of Problem 14-52, except that we ignore all minor losses (we set $\Sigma K_L = 0$). The resulting volume flow rate at the operating point is $\dot{V}$ **= 10,900 Lpm** (to three significant digits), approximately 54% higher than for the case with minor losses taken into account. In this problem, minor losses are not "minor", and are by no means negligible. Even though the duct is fairly long ($L/D$ is about 163), the minor losses are large, especially those through the damper and the one-way valve.

***Discussion*** An error of 54% is not acceptable in this type of problem. Furthermore, since it is not difficult to account for minor losses, especially if the calculations are performed on a computer, it is wise not to ignore these terms.

---

## 14-55

**Solution** We are to calculate pressure at two locations in a blocked duct system.

***Assumptions*** **1** The flow is steady. **2** The concentration of contaminants in the air is low; the fluid properties are those of air alone. **3** The air is at standard temperature and pressure (STP: 25°C and 101,300 Pa), and is incompressible.

***Properties*** The density of water at 25°C is 997.0 kg/m$^3$.

***Analysis*** Since the air is completely blocked by the one-way valve, there is no flow. Thus, there are no major or minor losses – just a pressure gain across the fan. Furthermore, the fan is operating at its shutoff head conditions. Since the pressure in the room is atmospheric, the gage pressure anywhere in the stagnant air region in the duct between the fan and the one-way valve is therefore equal to $H_0$ = 60.0 mm of water column. We convert to pascals as follows:

*Gage pressure at both locations*:
$$P_{\text{gage}} = \rho_{\text{water}} g H_0 = \left(998.0\ \frac{\text{kg}}{\text{m}^3}\right)\left(9.81\ \frac{\text{m}}{\text{s}^2}\right)(0.060\ \text{m})\left(\frac{\text{N}\cdot\text{s}^2}{\text{kg}\cdot\text{m}}\right)\left(\frac{\text{Pa}\cdot\text{m}^2}{\text{N}}\right) = 587\ \text{Pa}$$

Thus, at either location, the gage pressure is **60.0 mm of water column, or 587 Pa**.

***Discussion*** The answer depends only on the shutoff head of the fan – duct diameter, minor losses, etc, are irrelevant for this case since there is no flow. The fan should not be run for long time periods under these conditions, or it may burn out.

---

## 14-56E

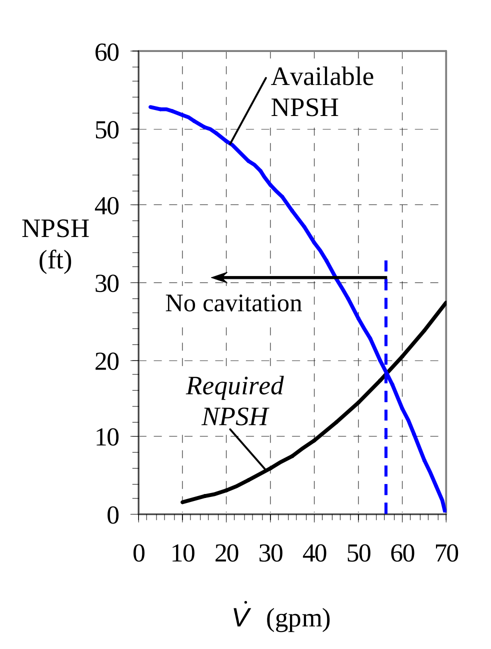
**Solution** For a given pump and piping system we are to estimate the maximum volume flow rate that can be pumped without cavitation.

***Assumptions*** **1** The flow is steady. **2** The water is incompressible. **3** The flow is turbulent.

***Properties*** $P_{atm}$ = 14.696 psi = 2116.2 lbf/ft$^2$. For water at $T$ = 77°F, $\mu = 6.002 \times 10^{-4}$ lbm/(ft·s), $\rho$ = 62.24 lbm/ft$^3$, and $P_v$ = 66.19 lbf/ft$^2$.

***Analysis*** We apply the steady energy equation in head form along a streamline from point 1 at the reservoir surface to point 2 at the pump inlet,

$$\frac{P_1}{\rho g} + \cancel{\frac{\alpha_1 V_1^2}{2g}} + z_1 + \cancel{h_{\text{pump,u}}} = \frac{P_2}{\rho g} + \frac{\alpha_2 V_2^2}{2g} + z_2 + \cancel{h_{\text{turbine}}} + h_{L,\text{total}} \qquad (1)$$




In Eq. 1 we have ignored the water speed at the reservoir surface ($V_1 \approx 0$). There is no turbine in the piping system. Also, although there is a pump in the system, there is no pump between point 1 and point 2; hence the pump head term also drops out. We solve Eq. 1 for $P_2/(\rho g)$, which is the pump inlet pressure expressed as a head,

$$\frac{P_2}{\rho g} = \frac{P_{\text{atm}}}{\rho g} + (z_1 - z_2) - \frac{\alpha_2 V_2^2}{2g} - h_{L,\text{total}} \qquad (2)$$

Note that in Eq. 2, we have recognized that $P_1 = P_{atm}$ since the reservoir surface is exposed to atmospheric pressure.

The available net positive suction head at the pump inlet is obtained from Eq. 14-8. After substitution of Eq. 2, approximating $\alpha_2 \approx 1$, we get an expression for the *available* NPSH:

$$\text{NPSH} = \frac{P_{\text{atm}} - P_v}{\rho g} + (z_1 - z_2) - h_{L,\text{total}} \qquad (3)$$

Since we know $P_{atm}$, $P_v$, and the elevation difference, all that remains is to estimate the total irreversible head loss through the piping system from the reservoir surface (1) to the pipe inlet (2), which depends on volume flow rate. Since the pipe diameter is constant, the total irreversible head loss becomes

$$h_{L,\text{total}} = \left( f\frac{L}{D} + \sum K_L \right)\frac{V^2}{2g} \qquad (4)$$

The rest of the problem is most easily solved on a computer. For a given volume flow rate, we calculate speed $V$ and Reynolds number Re. From Re and the known pipe roughness, we use the Moody chart (or the Colebrook equation) to obtain friction factor $f$. The sum of all the minor loss coefficients is

$$\sum K_L = 0.5 + 0.3 + 6.0 = 6.8 \qquad (5)$$

where we have not included the minor losses downstream of the pump, since they are irrelevant to the present analysis.

We make one calculation by hand for illustrative purposes. At $\dot{V}$ = 40.0 gpm, the average speed of water through the pipe is

$$V = \frac{\dot{V}}{A} = \frac{4\dot{V}}{\pi D^2} = \frac{4(40.0\text{ gal/min})}{\pi(1.2\text{ in})^2}\left(\frac{231\text{ in}^3}{\text{gal}}\right)\left(\frac{1\text{ min}}{60\text{ s}}\right)\left(\frac{1\text{ ft}}{12\text{ in}}\right) = 11.35\text{ ft/s} \qquad (6)$$

which produces a Reynolds number of Re = $\rho VD/\mu = 1.17 \times 10^5$. At this Reynolds number, and with roughness factor $\varepsilon/D$ = 0, the Colebrook equation yields $f$ = 0.0174. After substitution of the given variables along with $f$, $D$, $L$, and Eqs. 4, 5, and 6 into Eq. 3, we get

$$\text{NPSH} = \frac{(2116.2 - 66.19)\ \text{lbf/ft}^2}{(62.24\ \text{lbm/ft}^3)(32.174\ \text{ft/s}^2)}\left(\frac{32.174\ \text{lbm ft}}{\text{s}^2\text{lbf}}\right) + 20.0\ \text{ft} - \left(0.0174\frac{12.0\ \text{ft}}{0.10\ \text{ft}} + 6.8\right)\frac{(11.35\ \text{ft/s})^2}{2(32.174\ \text{ft/s}^2)} = 35.1\ \text{ft} \qquad (7)$$

The required net positive suction head is obtained from the given expression. At our example flow rate of 40.0 gpm we see that $\text{NPSH}_{\text{required}}$ is about 9.6 ft. Since the actual NPSH is much higher than this, we need not worry about cavitation at this flow rate. We use a spreadsheet to calculate NPSH as a function of volume flow rate, and the results are plotted. It is clear from the plot that **cavitation occurs at flow rates above about 56 gallons per minute**.

***Discussion*** $\text{NPSH}_{\text{required}}$ rises with volume flow rate, but the actual or available NPSH decreases with volume flow rate.

---

## 14-57E

**Solution** We are to calculate the volume flow rate below which cavitation in a pump is avoided.

***Assumptions*** **1** The flow is steady. **2** The water is incompressible.

***Properties*** $P_{\text{atm}}$ = 14.696 psi = 2116.2 lbf/ft$^2$. For water at $T$ = 150$^\circ$F, $\mu$ = 2.889 × 10$^{-4}$ lbm/ft·s, $\rho$ = 61.19 lbm/ft$^3$, and $P_v$ = 536.0 lbf/ft$^2$.

***Analysis*** The procedure is identical to that of the previous problem, except for the water properties. The calculations predict that **the pump cavitates at volume flow rates greater than about 53 gpm**. This is somewhat lower than the result of the previous problem, as expected, since cavitation occurs more readily in warmer water.

***Discussion*** Note that $\text{NPSH}_{\text{required}}$ does not depend on water temperature, but the actual or available NPSH decreases with temperature.

---

**14-58**

**Solution** For a given pump and piping system we are to estimate the maximum volume flow rate that can be pumped without cavitation.

***Assumptions*** **1** The flow is steady. **2** The water is incompressible. **3** The flow is turbulent.

***Properties*** Standard atmospheric pressure is $P_{atm}$ = 101.3 kPa. For water at $T$ = 25°C, $\rho$ = 997.0 kg/m³, $\mu = 8.91\times10^{-4}$ kg/m·s, and $P_v$ = 3.169 kPa.

***Analysis*** We apply the steady energy equation in head form along a streamline from point 1 at the reservoir surface to point 2 at the pump inlet,

*Energy equation*:
$$\frac{P_1}{\rho g}+\frac{\alpha_1 V_1^2}{2g}+z_1+h_{\text{pump,u}}=\frac{P_2}{\rho g}+\frac{\alpha_2 V_2^2}{2g}+z_2+h_{\text{turbine}}+h_{L,\text{total}} \qquad (1)$$

In Eq. 1 we have ignored the water speed at the reservoir surface ($V_1 \approx 0$). There is no turbine in the piping system. Also, although there is a pump in the system, there is no pump between point 1 and point 2; hence the pump head term also drops out. We solve Eq. 1 for $P_2/(\rho g)$, which is the pump inlet pressure expressed as a head,

*Pump inlet pressure head*:
$$\frac{P_2}{\rho g}=\frac{P_{\text{atm}}}{\rho g}+\left(z_1-z_2\right)-\frac{\alpha_2 V_2^2}{2g}-h_{L,\text{total}} \qquad (2)$$

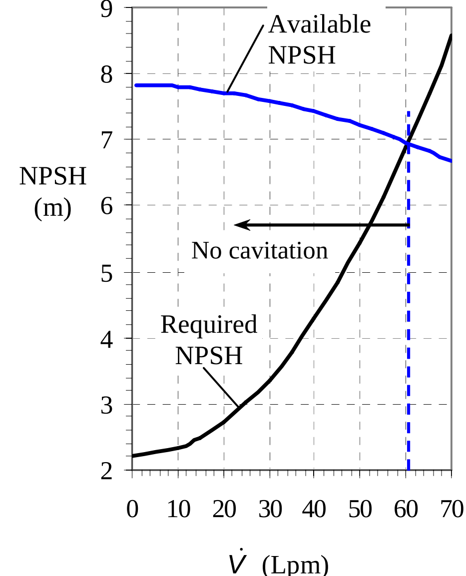
Note that in Eq. 2, we have recognized that $P_1 = P_{atm}$ since the reservoir surface is exposed to atmospheric pressure.

The available net positive suction head at the pump inlet is obtained from Eq. 14-8. After substitution of Eq. 2, and approximating $\alpha_2$ as 1.0, we get

*Available* NPSH:
$$\text{NPSH}=\frac{P_{\text{atm}}-P_v}{\rho g}+\left(z_1-z_2\right)-h_{L,\text{total}} \qquad (3)$$

Since we know $P_{atm}$, $P_v$, and the elevation difference, all that remains is to estimate the total irreversible head loss through the piping system from the reservoir surface (1) to the pipe inlet (2), which depends on volume flow rate. Since the pipe diameter is constant, the total irreversible head loss is

$$h_{L,\text{total}}=\left(f\frac{L}{D}+\sum K_L\right)\frac{V^2}{2g} \qquad (4)$$




The rest of the problem is most easily solved on a computer. For a given volume flow rate, we calculate speed $V$ and Reynolds number Re. From Re and the known pipe roughness, we use the Moody chart (or the Colebrook equation) to obtain friction factor $f$. The sum of all the minor loss coefficients is

$$\sum K_L = 0.85+0.3=1.15 \qquad (5)$$

where we have not included the minor losses downstream of the pump, since they are irrelevant to the present analysis.

We make one calculation by hand for illustrative purposes. At $\dot{V}$ = 40.0 Lpm, the average speed of water through the pipe is

$$V=\frac{\dot{V}}{A}=\frac{4\dot{V}}{\pi D^2}=\frac{4\left(40.0\text{ L/min}\right)}{\pi\left(0.024\text{ m}\right)^2}\left(\frac{1\text{ m}^3}{1000\text{ L}}\right)\left(\frac{1\text{ min}}{60\text{ s}}\right)=1.474\text{ m/s} \qquad (6)$$

which produces a Reynolds number of Re = $\rho VD/\mu = 3.96\times10^4$. At this Reynolds number, and with roughness factor $\varepsilon/D$ = 0, the Colebrook equation yields $f$ = 0.0220. After substitution of the given variables, along with $f$, $D$, $L$, and Eqs. 4, 5, and 6 into Eq. 3, we calculate the available NPSH,

$$\text{NPSH} = \frac{(101{,}300 - 3{,}169)\ \text{N/m}^2}{(997.0\ \text{kg/m}^3)(9.81\ \text{m/s}^2)}\left(\frac{\text{kg m}}{\text{s}^2\text{N}}\right) - 2.2\ \text{m} - \left(0.0220\frac{2.8\ \text{m}}{0.024\ \text{m}} + 1.15\right)\frac{(1.474\ \text{m/s})^2}{2(9.81\ \text{m/s}^2)} = 7.42\ \text{m} \qquad (7)$$

The required net positive suction head is obtained from the given expression. At our example flow rate of 40.0 Lpm we see that $\text{NPSH}_{\text{required}}$ is about 4.28 m. Since the actual NPSH is higher than this, the pump does not cavitate at this flow rate. We use a spreadsheet to calculate NPSH as a function of volume flow rate, and the results are plotted. It is clear from this plot that **cavitation occurs at flow rates above 60.5 liters per minute**.

***Discussion*** $\text{NPSH}_{\text{required}}$ rises with volume flow rate, but the actual or available NPSH decreases with volume flow rate.

---

## 14-59

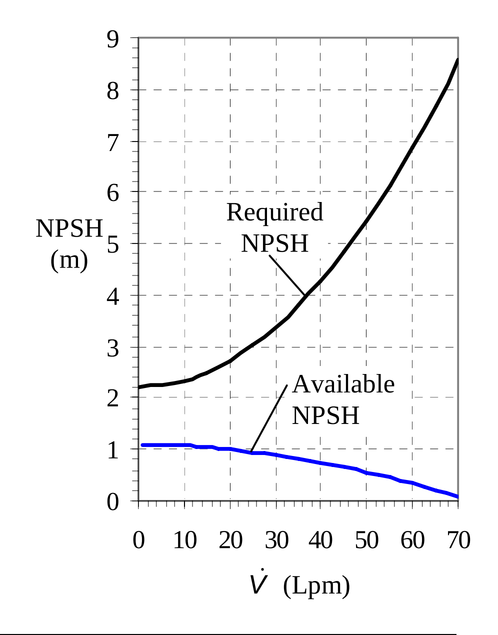
**Solution** We are to calculate the volume flow rate below which cavitation in a pump is avoided, at two temperatures.

***Assumptions*** **1** The flow is steady. **2** The water is incompressible.

***Properties*** Standard atmospheric pressure is $P_{\text{atm}}$ = 101.3 kPa. For water at $T$ = 80°C, $\rho$ = 971.9 kg/m$^3$, $\mu$ = 3.55×10$^{-4}$ kg/m·s, and $P_v$ = 47.35 kPa. At $T$ = 90°C, $\rho$ = 965.3 kg/m$^3$, $\mu$ = 3.15×10$^{-4}$ kg/m·s, and $P_v$ = 70.11 kPa.

***Analysis*** The procedure is identical to that of the previous problem, except for the water properties. The calculations predict that **at *T* = 80°C, the pump cavitates at volume flow rates greater than 28.0 Lpm**. This is substantially lower than the result of the previous problem, as expected, since cavitation occurs more readily in warmer water.

At 90°C, the vapor pressure is very high since the water is near boiling (at atmospheric pressure, water boils at 100°C). For this case, the curves of $\text{NPSH}_{\text{available}}$ and $\text{NPSH}_{\text{required}}$ do not cross at all as seen in the plot, implying that **the pump cavitates at any flow rate when *T* = 90°C**.




***Discussion*** Note that $\text{NPSH}_{\text{required}}$ does not depend on water temperature, but the actual or available NPSH decreases with temperature.

---

## 14-60

**Solution** We are to calculate the volume flow rate below which cavitation in a pump is avoided, and compare with a previous case with a smaller pipe diameter.

***Assumptions*** **1** The flow is steady. **2** The water is incompressible.

***Properties*** Standard atmospheric pressure is $P_{\text{atm}}$ = 101.3 kPa. For water at $T$ = 25°C, $\rho$ = 997.0 kg/m$^3$, $\mu$ = 8.91×10$^{-4}$ kg/m·s, and $P_v$ = 3.169 kPa.

***Analysis*** The analysis is identical to that of Problem 14-58, except that the pipe diameter is 48.0 mm instead of 24.0 mm. Compared to problem 14-58, at a given volume flow rate, the average speed through the pipe decreases by a factor of four since the pipe area increases by a factor of four. The Reynolds number goes down by a factor of two, but the flow is still turbulent (At our sample flow rate of 40.0 Lpm, Re = 1.98 × 10$^4$). For a smooth pipe at this Reynolds number, $f$ = 0.0260. and the available NPSH is 7.81 m, slightly higher than the previous case with the smaller diameter pipe. After repeating the calculations at several flow rates, we find that **the pump cavitates at $\dot{V}$ = 65.5 Lpm**. This represents an increase of about 8.3%. **Cavitation occurs at a higher volume flow rate when the pipe diameter is increased because the irreversible head losses in the piping system upstream of the pump are decreased**.

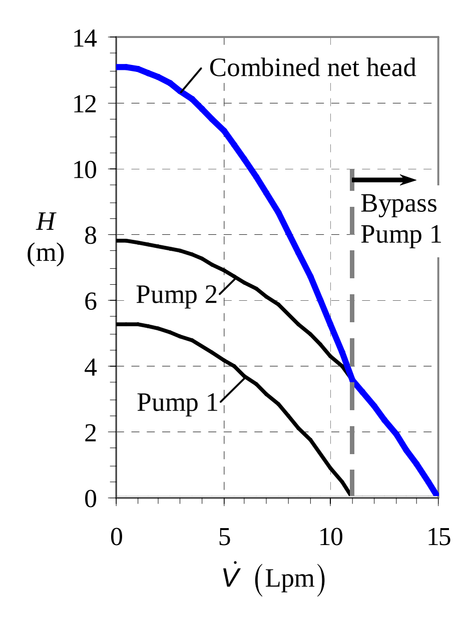
***Discussion*** If a computer program like EES was used for Problem 14-58, it is a trivial matter to change the pipe diameter and re-do the calculations.

---

## 14-61

**Solution** We are to calculate the combined shutoff head and free delivery for two pumps in series, and discuss why the weaker pump should be shut off and bypassed above some flow rate.

***Assumptions*** **1** The water is incompressible. **2** The flow is steady.




***Analysis*** The pump performance curves for both pumps individually and for their combination in series are plotted in Fig. 1. At zero flow rate, the shutoff head of the two pumps in series is equal to the sum of their individual shutoff heads: $H_{0,\text{combined}} = H_{0,1} + H_{0,2} = 5.30$ m + 7.80 m. Thus **the combined shutoff head is 13.1 m**. At free delivery conditions (zero net head), the free delivery of the two pumps in series is limited to that of the stronger pump, in this case Pump 2: $\dot{V}_{\text{max,combined}} = \text{MAX}(\dot{V}_{\text{max,1}}, \dot{V}_{\text{max,2}}) = 15.0$ Lpm. Thus **the combined free delivery is 15.0 Lpm**.

As volume flow rate increases, the combined net pump head is equal to the sum of the net pump heads of the individual pumps, as seen on the plot. However, the free delivery of Pump 1 is

$$0 = H_0 - a\dot{V}^2 \quad \rightarrow \quad \dot{V}_{\max} = \sqrt{\frac{H_0}{a}} = \sqrt{\frac{5.30\text{ m}}{0.0438\text{ m/(Lpm)}^2}} = 11.0\text{ Lpm}$$

Above this flow rate, Pump 1 no longer contributes to the flow. In fact, it becomes a liability to the system since its net head would be *negative* at flow rates above 11.0 Lpm. For this reason, **it is wise to shut off and bypass Pump 1 above 11.0 Lpm**.

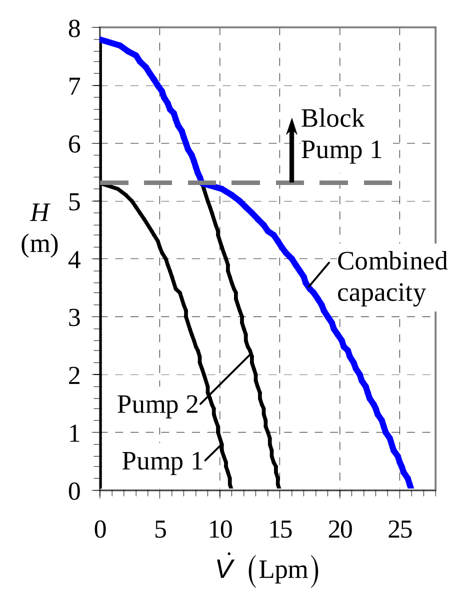
***Discussion*** The free delivery of Pump 2 is 15.0 Lpm. The free delivery of the two-pump system is no better than that of Pump 2 alone, since Pump 1 is shut off and bypassed at flow rates above 11.0 Lpm.

---

## 14-62

**Solution** We are to calculate the free delivery and shutoff head for two pumps in parallel, and discuss why the weaker pump should be shut off and bypassed above some net head.

***Assumptions*** **1** The water is incompressible. **2** The flow is steady.




***Analysis*** The pump performance curves for both pumps and for their combination in parallel are plotted. At zero flow rate, the shutoff head of the two pumps in parallel is equal to that of the stronger pump, in this case Pump 2: $H_{0,\text{combined}} = \text{MAX}(H_{0,1}, H_{0,2}) = 7.80$ m. Thus **the combined shutoff head is 7.80 m**. At free delivery conditions (zero net head), the free delivery of the two pumps in parallel is the sum of their individual free deliveries: $\dot{V}_{\text{max,combined}} = \dot{V}_{\text{max,1}} + \dot{V}_{\text{max,2}} = 11.0$ Lpm + 15.0 Lpm. Thus **the combined free delivery is 26.0 Lpm**.

As net head increases, the combined capacity is equal to the sum of the capacities of the individual pumps, as seen on Fig. 1. However, the shutoff head of Pump 1 (5.30 m) is lower than that of Pump 2 (7.80 m). Above the shutoff head of Pump 1, that pump no longer contributes to the flow. In fact, it becomes a liability to the system since it cannot sustain such a high head. If not shut off and blocked, the volume flow rate through Pump 1 would be *negative* at net heads above 7.80 m. For this reason, **it is wise to shut off and block Pump 1 above 7.80 m**.

***Discussion*** The shutoff head of Pump 2 is 7.80 m. The shutoff head of the two-pump system is no better than that of Pump 2 alone, since Pump 1 is shut off and blocked at net heads above 7.80 m.

---

## 14-63E

**Solution** We are to calculate the mass flow rate of slurry through a two-lobe rotary pump for given values of lobe volume and rotation rate.

***Assumptions*** **1** The flow is steady in the mean. **2** There are no leaks in the gaps between lobes or between lobes and the casing. **3** The slurry is incompressible.

***Analysis*** By studying the figure provided with this problem, we see that for each 360° rotation of the two counter-rotating shafts ($n$ = 1 rotation), the total volume of pumped fluid is

*Closed volume pumped per rotation*: $$V_{closed} = 4V_{lobe} \qquad (1)$$

The volume flow rate is then calculated from Eq. 14-11,

$$\dot{V} = \dot{n}\frac{V_{closed}}{n} = (300 \text{ rot/min})\frac{4(0.145 \text{ gal})}{1 \text{ rot}} = 174 \text{ gal/min} \qquad (2)$$

Thus, **the volume flow rate is 174 gpm**.

***Discussion*** If there were leaks in the pump, the volume flow rate would be lower. The fluid density was not used in the calculation of volume flow rate. However, the higher the density, the higher the required shaft torque and brake horsepower.

---

## 14-64E

**Solution** We are to calculate the mass flow rate of slurry through a three-lobe rotary pump for given values of lobe volume and rotation rate.

***Assumptions*** **1** The flow is steady in the mean. **2** There are no leaks in the gaps between lobes or between lobes and the casing. **3** The slurry is incompressible.

***Analysis*** When there are three lobes, three lobe volumes are pumped for each 360° rotation of each rotor ($n$ = 1 rotation). Thus, the total volume of pumped fluid is

*Closed volume pumped per rotation*: $$V_{closed} = 6V_{lobe} \qquad (1)$$

The volume flow rate is then calculated from Eq. 14-11,

$$\dot{V} = \dot{n}\frac{V_{closed}}{n} = (300 \text{ rot/min})\frac{6(0.087 \text{ gal})}{1 \text{ rot}} = 157 \text{ gal/min} \qquad (2)$$

Thus, **the volume flow rate is 157 gpm**.

***Discussion*** If there were leaks in the pump, the volume flow rate would be lower. This flow rate is slightly lower than that of the previous problem. Why? For the same overall diameter, it is clear from geometry that the more lobes, the less the volume per lobe, and the more "wasted" volume inside the pump.

---

**14-65**

**Solution** We are to calculate the volume flow rate of tomato paste through a positive-displacement pump for given values of lobe volume and rotation rate.

***Assumptions*** **1** The flow is steady in the mean. **2** There are no leaks in the gaps between lobes or between lobes and the casing. **3** The fluid is incompressible.

***Analysis*** By studying Fig. 14-27 or 14-30, we see that for each 360$^{o}$ rotation of the two counter-rotating shafts ($n$ = 1 rotation), the total volume of pumped fluid is

*Closed volume pumped per rotation*: $$V_{\text{closed}} = 4V_{\text{lobe}} \qquad (1)$$

The volume flow rate is then calculated from Eq. 14-11,

$$\dot{V} = \dot{n}\frac{V_{\text{closed}}}{n} = \left(400 \text{ rot/min}\right)\frac{4\left(3.64 \text{ cm}^3\right)}{1 \text{ rot}} = 5820 \text{ cm}^3\text{/min} \qquad (2)$$

Thus, **the volume flow rate is 5820 cm$^3$/min**.

***Discussion*** If there were leaks in the pump, the volume flow rate would be lower. The fluid density was not used in the calculation of volume flow rate. However, the higher the fluid density, the higher the required shaft torque and brake horsepower.

---

**14-66**

**Solution** We are to calculate the volume flow rate per rotation of a gear pump.

***Assumptions*** **1** The flow is steady in the mean. **2** There are no leaks in the gaps between lobes or between lobes and the casing. **3** The fluid is incompressible.

***Analysis*** From Fig. 14-26c, we count 14 teeth per gear. Thus, for each 360$^{o}$ rotation of each gear ($n$ = 1 rotation), 14·(0.350 cm$^3$) of fluid is pumped. Sine there are two gears, the total volume of fluid pumped per rotation is 2(14)(0.350 cm$^3$) = **9.80 cm$^3$**.

***Discussion*** The actual volume flow rate will be lower than this due to leakage in the gaps.

---

**14-67**

**Solution** We are to calculate the brake horsepower and net head of an idealized centrifugal pump at a given volume flow rate and rotation rate.

***Assumptions*** **1** The flow is steady in the mean. **2** The water is incompressible. **3** The efficiency of the pump is 100% (no irreversible losses).

***Properties*** We take the density of water to be $\rho$ = 998.0 kg/m$^3$.

***Analysis*** Since the volume flow rate (capacity) is given, we calculate the normal velocity components at the inlet and the outlet using Eq. 14-12,

*Normal velocity, inlet*:

$$V_{1,\mathrm{n}} = \frac{\dot{V}}{2\pi r_1 b_1} = \frac{0.573\ \mathrm{m^3/s}}{2\pi\left(0.120\ \mathrm{m}\right)\left(0.180\ \mathrm{m}\right)} = 4.222\ \mathrm{m/s}$$

$V_1 = V_{1,\mathrm{n}}$, and $V_{1,\mathrm{t}} = 0$, since $\alpha_1 = 0^{\mathrm{o}}$. Similarly, $V_{2,\mathrm{n}}$ = 2.714 m/s and

*Tangential component of absolute velocity at impeller outlet*:

$$V_{2,\mathrm{t}} = V_{2,\mathrm{n}} \tan\alpha_2 = \left(2.714\ \mathrm{m/s}\right)\tan(35^{\mathrm{o}}) = 1.900\ \mathrm{m/s}$$

Now we use Eq. 14-17 to predict the net head,

$$H = \frac{\omega}{g}\left(r_2 V_{2,\mathrm{t}} - r_1 \underbrace{V_{1,\mathrm{t}}}_{0}\right) = \frac{78.54\ \mathrm{rad/s}}{9.81\ \mathrm{m/s^2}}\left(0.240\ \mathrm{m}\right)\left(1.900\ \mathrm{m/s}\right) = \mathbf{3.65\ m}$$

Finally, we use Eq. 14-16 to predict the required brake horsepower,

$$bhp = \rho g \dot{V} H = \left(998.\ \mathrm{kg/m^3}\right)\left(9.81\ \mathrm{m/s^2}\right)\left(0.573\ \mathrm{m^3/s}\right)\left(3.65\ \mathrm{m}\right)\left(\frac{\mathrm{W\cdot s^3}}{\mathrm{kg\cdot m^2}}\right) = \mathbf{20{,}500\ W}$$

***Discussion*** The actual net head delivered to the water will be lower than this due to inefficiencies. Similarly, actual brake horsepower will be higher than that predicted here due to inefficiencies in the pump, friction on the shaft, etc.

**14-68**

**Solution** For given flow conditions and stator blade shape at a given radius, we are to design the rotor blade. Specifically, we are to calculate the leading and trailing edge angles of the rotor blade and sketch its shape. We are also to decide how many rotor blades to construct.

***Assumptions*** **1** The air is nearly incompressible. **2** The flow area between hub and tip is constant. **3** Two-dimensional blade row analysis is appropriate.

***Analysis*** First we analyze flow through the stator from an absolute reference frame, using the two-dimensional approximation of a cascade (blade row) of stator blades as sketched in Fig. 1. Flow enters axially (horizontally), and is turned 26.6° downward. Since the axial component of velocity must remain constant to conserve mass, the magnitude of the velocity leaving the trailing edge of the stator, $\vec{V}_{st}$ is calculated,

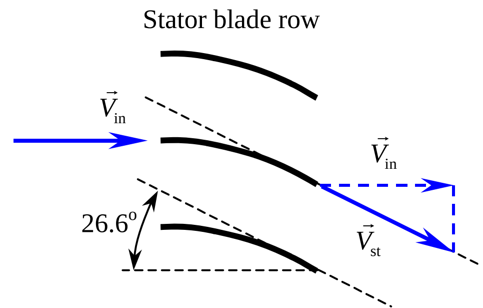



**FIGURE 1**
Analysis of the stator of a vaneaxial flow fan as a two-dimensional cascade of stator blades; absolute reference frame.

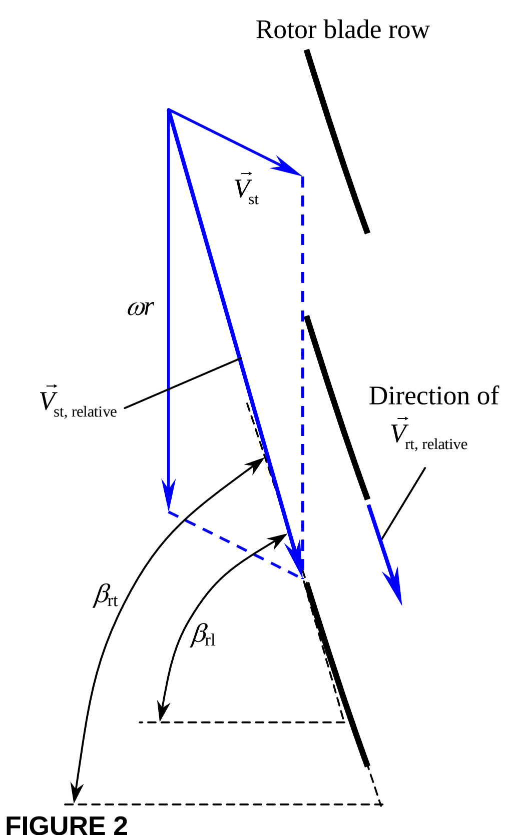



**FIGURE 2**
Analysis of the stator trailing edge velocity as it impinges on the rotor leading edge; relative reference frame.

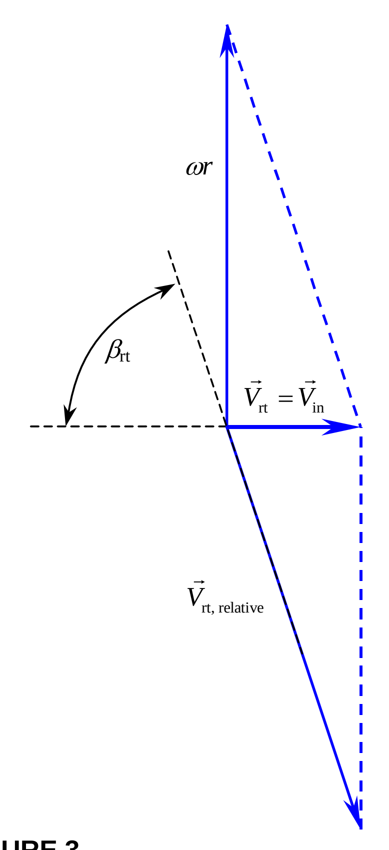



**FIGURE 3**
Analysis of the rotor trailing edge velocity; absolute reference frame.

$$V_{st} = \frac{V_{in}}{\cos\beta_{st}} = \frac{31.4 \text{ m/s}}{\cos\left(26.6^\circ\right)} = 35.12 \text{ m/s} \qquad (1)$$

The direction of $\vec{V}_{st}$ is assumed to be that of the rotor trailing edge. In other words we assume that the flow turns nicely through the blade row and exits parallel to the trailing edge of the blade, as shown in the sketch.

We convert $\vec{V}_{st}$ to the *relative* reference frame moving with the rotor blades. At a radius of 0.50 m, the tangential velocity of the rotor blades is

$$u_\theta = \omega r = \left[\left(1800 \text{ rot/min}\right)\left(\frac{2\pi \text{ rad}}{\text{rot}}\right)\left(\frac{1 \text{ min}}{60 \text{ s}}\right)\right]\left(0.50 \text{ m}\right) = 94.25 \text{ m/s} \qquad (2)$$

Since the rotor blade row moves upward in the figure provided with this problem, we add a *downward* velocity with magnitude given by Eq. 2 to translate $\vec{V}_{\text{st}}$ into the rotating reference frame sketched in Fig. 2. The angle of the leading edge of the rotor, $\beta_{\text{rl}}$, can be calculated. After some trig,

$$\beta_{\text{rl}} = \arctan\frac{\omega r + V_{\text{in}} \tan\beta_{\text{st}}}{V_{\text{in}}} = \arctan\frac{94.25 \text{ m/s} + \left(31.4 \text{ m/s}\right)\tan\left(26.6^{\circ}\right)}{31.4 \text{ m/s}} = 74.06^{\circ} \qquad (3)$$

The air must now be turned by the rotor blade row in such a way that it leaves the trailing edge of the rotor blade at zero angle (axially – no swirl) from an absolute reference frame. This determines the rotor's trailing edge angle, $\beta_{\text{rt}}$. Specifically, when we add an *upward* velocity of magnitude $\omega r$ (Eq. 2) to the relative velocity exiting the trailing edge of the rotor, $\vec{V}_{\text{rt, relative}}$, we convert back to the absolute reference frame, and obtain $\vec{V}_{\text{rt}}$, the velocity leaving the rotor trailing edge. It is this velocity, $\vec{V}_{\text{rt}}$, which must be axial (horizontal). Furthermore, to conserve mass $\vec{V}_{\text{rt}}$ must equal $\vec{V}_{\text{in}}$ since we are assuming incompressible flow. Working "backwards" we construct $\vec{V}_{\text{rt, relative}}$ in Fig. 3. Some trigonometry reveals that

$$\beta_{\text{rt}} = \arctan\frac{\omega r}{V_{\text{in}}} = \arctan\frac{94.25 \text{ m/s}}{31.4 \text{ m/s}} = 71.57^{\circ} \qquad (4)$$

We conclude that the rotor blade at this radius has a leading edge angle of about **74.1°** (Eq. 3) and a trailing edge angle of about **71.6°** (Eq. 4). A sketch of the rotor blade is provided in Fig. 2; it is clear that the blade is nearly straight, at least at this radius.

Finally, to avoid interaction of the stator blade wakes with the rotor blade leading edges, we choose the number of rotor blades such that it has no common denominator with the number of stator blades. Since there are 18 stator blades, we pick a number like **13**, **17**, or **19** rotor blades. 16 would not be appropriate since it shares a common denominator of 2 with the number 18.

***Discussion*** We can easily repeat the calculation for all radii from hub to tip, completing the design of the entire rotor.

## Turbines

### 14-69C

**Solution** We are to discuss why turbines have higher efficiencies than pumps.

***Analysis*** There are several reasons for this. First, **pumps normally operate at higher rotational speeds than do turbines; therefore, shear stresses and frictional losses are higher**. Second, **conversion of kinetic energy into flow energy (pumps) has inherently higher losses than does the reverse (turbines)**. You can think of it this way: Since pressure *rises* across a pump (adverse pressure gradient), but *drops* across a turbine (favorable pressure gradient), boundary layers are less likely to separate in a turbine than in a pump. Third, **turbines (especially hydroturbines) are often much larger than pumps, and viscous losses become less important as size increases**. Finally, while pumps often operate over a wide range of flow rates, **most electricity-generating turbines run within a narrower operating range and at a controlled constant speed**; they can therefore be designed to operate very efficiently at those conditions.

***Discussion*** Students' answers should be in their own words.

### 14-70C

**Solution** We are to discuss the classification of dynamic pumps and reaction turbines.

***Analysis*** **Dynamic pumps are classified according to the angle at which the flow *exits* the impeller blade** – centrifugal, mixed-flow, or axial. **Reaction turbines, on the other hand, are classified according to the angle that the flow *enters* the runner** – radial, mixed-flow, or axial. This is the main difference between how dynamic pumps and reaction turbines are classified.

***Discussion*** Students' answers should be in their own words.

### 14-71C

**Solution** We are to discuss the meaning and purpose of draft tubes.

***Analysis*** **A draft tube is a diffuser that also turns the flow downstream of a turbine. Its purpose is to turn the flow horizontally and recover some of the kinetic energy leaving the turbine runner. If the draft tube is not designed carefully, much of the kinetic energy leaving the runner would be wasted, reducing the overall efficiency of the turbine system**.

***Discussion*** Students' answers should be in their own words.

### 14-72C

**Solution** We are to name and describe the two types of dynamic turbine.

***Analysis*** There are two basic types of dynamic turbine – **impulse** and **reaction**. In an impulse turbine, fluid is sent through a nozzle so that most of its available mechanical energy is converted into kinetic energy. The high-speed jet then impinges on bucket-shaped vanes that transfer energy to the turbine shaft. In a reaction turbine, the fluid completely fills the casing, and the runner is rotated by momentum exchange due to pressure differences across the blades, rather than by kinetic energy impingement. Impulse turbines require a higher head, but can operate with a smaller volume flow rate. Reaction turbines can operate with much less head, but require higher volume flow rate.

***Discussion*** Students' answers should be in their own words.

## 14-73C

**Solution** We are to discuss reverse swirl in reaction turbines.

***Analysis*** **Reverse swirl is when the runner blades turn the flow so much that the swirl at the runner outlet is in the direction opposite to runner rotation**. Reverse swirl is desirable so that more power is absorbed from the water. We can easily see this from the Euler turbomachine equation,

$$\dot{W}_{\text{shaft}} = \omega \text{T}_{\text{shaft}} = \rho\omega\dot{V}\left(r_2V_{2,\text{t}} - r_1V_{1,\text{t}}\right) \qquad (1)$$

Namely, since there is a negative sign on the last term, the shaft power increases if $V_{1,\text{t}}$ is *negative*, in other words, if there is reverse swirl at the runner outlet. If there is too much reverse swirl, a lot of extra kinetic energy gets wasted downstream of the runner.

***Discussion*** A well designed draft tube can recover a good portion of the streamwise kinetic energy of the water leaving the runner. However, the swirling kinetic energy cannot be recovered.

## 14-74

**Solution** We are to prove that the maximum power of a Pelton wheel occurs when $\omega r = V_j/2$ (bucket moving at half the jet speed).

***Assumptions*** **1** Frictional losses are negligible, so that the Euler turbomachine equation applies, and the relative exit speed of the jet is the same as its relative inlet speed. **2** The turning angle is sufficient to prevent the exiting fluid from striking the next bucket. **3** The fluid, jet speed, volume flow rate, turning angle, and wheel radius are fixed – rotation rate is the only variable about which we are concerned.

***Analysis*** With the stated assumptions, Eq. 14-40 applies, namely,

*Output shaft power*:

$$\dot{W}_{\text{shaft}} = \rho\omega r\dot{V}\left(V_j - \omega r\right)\left(1 - \cos\beta\right) \qquad (1)$$

We differentiate Eq. 1 with respect to $\omega$, and set the derivative equal to zero,

*Maximum power*:

$$\frac{d\dot{W}_{\text{shaft}}}{d\omega} = 0 \quad \rightarrow \quad \frac{d}{d\omega}\left(\omega V_j - \omega^2 r\right) = 0 \quad \rightarrow \quad V_j - 2\omega r = 0 \qquad (2)$$

Solution of Eq. 2 yields the desired result, namely, the maximum power of a Pelton wheel occurs when $\boldsymbol{\omega r = V_j/2}$ (bucket moving at half the jet speed).

***Discussion*** To be sure that we have not identified a *minimum* instead of a maximum, we could substitute some numerical values and plot $\dot{W}_{\text{shaft}}$ versus $\omega$. It turns out that we have indeed found the maximum value of $\dot{W}_{\text{shaft}}$.

## 14-75

**Solution** We are to calculate several performance parameters for a Pelton wheel turbine.

***Assumptions*** **1** Frictional losses are negligible, so that the Euler turbomachine equation applies, and the relative exit speed of the jet is the same as its relative inlet speed. **2** The turning angle is sufficient to prevent the exiting fluid from striking the next bucket. **3** The water is at 20°C.

***Properties*** The density of water at $T$ = 20°C is 998.0 kg/m$^3$.

***Analysis*** ($a$) The volume flow rate of the jet is equal to jet area times jet velocity,

$$\dot{V} = V_j \pi D_j^2 / 4 = (102 \text{ m/s})\pi(0.100 \text{ m})^2 / 4 = \mathbf{0.801 \, m^3/s}$$

($b$) The maximum output shaft power occurs when the bucket moves at half the jet speed ($\omega r = V_j/2$). Thus,

$$\omega = \frac{V_j}{2r} = \frac{102 \text{ m/s}}{2(1.83 \text{ m})} = 27.87 \text{ rad/s}$$

$$\rightarrow \quad \dot{n} = (27.87 \text{ rad/s})\left(\frac{\text{rot}}{2\pi \text{ rad}}\right)\left(\frac{60 \text{ s}}{\text{min}}\right) = \mathbf{266 \, rpm}$$

($c$) The ideal shaft power is found from Eq. 14-40,

$$\dot{W}_{\text{ideal}} = \rho \omega r \dot{V}(V_j - \omega r)(1 - \cos\beta)$$

$$= (998.0 \text{ kg/m}^3)\left(27.87 \frac{\text{rad}}{\text{s}}\right)(1.83 \text{ m})\left(0.801 \frac{\text{m}^3}{\text{s}}\right)\left(\frac{102 \text{ m/s}}{2}\right)(1 - \cos 165^\circ)$$

$$\times \left(\frac{\text{N} \cdot \text{s}^2}{\text{kg} \cdot \text{m}}\right)\left(\frac{\text{W} \cdot \text{s}}{\text{N} \cdot \text{m}}\right)\left(\frac{1 \text{ MW}}{10^6 \text{ W}}\right) = 4.09 \text{ MW}$$

where we have substituted $\omega r = V_j/2$ for convenience in the calculations. Since the turbine efficiency is given, we calculate the actual output shaft power, or brake horsepower,

$$\dot{W}_{\text{actual}} = bhp = \dot{W}_{\text{ideal}} \eta_{\text{turbine}} = (4.09 \text{ MW})(0.82) = \mathbf{3.35 \, MW}$$

***Discussion*** At other rotation speeds, the turbine would not be as efficient.

---

## 14-76

**Solution** We are to estimate the ideal power production from a hydroturbine.

***Assumptions*** **1** Frictional losses are negligible. **2** The water is at 20°C.

***Properties*** The density of water at $T$ = 20°C is 998.0 kg/m$^3$.

***Analysis*** The ideal power produced by a hydroturbine is

$$\dot{W}_{\text{ideal}} = \rho g \dot{V} H_{\text{gross}} = (998.0 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(1.5 \text{ m}^3\text{/s})(650 \text{ m}) \times \left(\frac{\text{N} \cdot \text{s}^2}{\text{kg} \cdot \text{m}}\right)\left(\frac{\text{W} \cdot \text{s}}{\text{N} \cdot \text{m}}\right)\left(\frac{1 \text{ MW}}{10^6 \text{ W}}\right) = \mathbf{9.55 \, MW}$$

***Discussion*** If a hydroelectric dam were to be built on this site, the actual power output per turbine would be smaller than this, of course, due to inefficiencies.

---

**14-77E**

**Solution** We are to estimate the power production from a hydroelectric plant.

***Properties*** The density of water at $T$ = 70°F is 62.30 lbm/ft³.

***Analysis*** The ideal power produced by one hydroturbine is

$$\dot{W}_{\text{ideal}} = \rho g \dot{V} H_{\text{gross}}$$

$$= \left(62.30\text{ lbm/ft}^3\right)\left(32.2\text{ ft/s}^2\right)\left(203{,}000\text{ gal/min}\right)\left(1065\text{ ft}\right)\times\left(\frac{\text{lbf}\cdot\text{s}^2}{32.2\text{ lbm}\cdot\text{ft}}\right)\left(0.1337\frac{\text{ft}^3}{\text{gal}}\right)\left(\frac{1.356\text{ W}}{\text{ft}\cdot\text{lbf/s}}\right)\left(\frac{1\text{ min}}{60\text{ s}}\right)\left(\frac{1\text{ MW}}{10^6\text{ W}}\right)$$

$$= 40.70\text{ MW}$$

But inefficiencies in the turbine, the generator, and the rest of the system reduce the actual electrical power output. For each turbine,

$$\dot{W}_{\text{electrical}} = \dot{W}_{\text{ideal}}\eta_{\text{turbine}}\eta_{\text{generator}}\eta_{\text{other}} = \left(40.70\text{ MW}\right)\left(0.952\right)\left(0.945\right)\left(1-0.035\right) = 35.3\text{ MW}$$

Finally, since there are 12 turbines in parallel, the total power produced is

$$\dot{W}_{\text{total, electrical}} = 12\dot{W}_{\text{electrical}} = 12\left(35.3\text{ MW}\right) = \mathbf{424\text{ MW}}$$

***Discussion*** A small improvement in any of the efficiencies ends up increasing the power output, and increases the power company's profitability.

**14-78**

**Solution** We are to calculate runner blade angles, required net head, and power output for a Francis turbine.

***Assumptions*** **1** The flow is steady. **2** The fluid is water at 20°C. **3** The blades are infinitesimally thin. **4** The flow is everywhere tangent to the runner blades. **5** We neglect irreversible losses through the turbine.

***Properties*** For water at 20°C, $\rho$ = 998.0 kg/m$^3$.

***Analysis*** We solve for the normal component of velocity at the inlet,

$$V_{2,n} = \frac{\dot{V}}{2\pi r_2 b_2} = \frac{340 \text{ m}^3/\text{s}}{2\pi(2.00 \text{ m})(0.731 \text{ m})} = 37.0 \text{ m/s} \quad (1)$$

Using the figure provided with this problem as a guide, the tangential velocity component at the inlet is

$$V_{2,t} = V_{2,n}\tan\alpha_2 = (37.0 \text{ m/s})\tan 30^\circ = 21.4 \text{ m/s} \quad (2)$$

The angular velocity is $\omega = 2\pi\dot{n}/60$ = 18.85 rad/s.

The tangential velocity component of the absolute velocity at the inlet is obtained from trigonometry to be (see Eq. 14-45)

$$V_{2,t} = \omega r_2 - \frac{V_{2,n}}{\tan\beta_2}$$

From the above relationship, we solve for the runner leading edge angle $\beta_2$,

$$\beta_2 = \arctan\left[\frac{V_{2,n}}{\omega r_2 - V_{2,t}}\right] = \arctan\left[\frac{37.0 \text{ m/s}}{(18.85 \text{ rad/s})(2.00 \text{ m}) - 21.4 \text{ m/s}}\right] = \mathbf{66.2^\circ} \quad (3)$$

Equations 1 through 3 are repeated for the runner outlet, with the following results:

*Runner outlet*: $V_{1,n} = 17.3 \text{ m/s}, \quad V_{1,t} = 3.05 \text{ m/s}, \quad \beta_1 = \mathbf{36.1^\circ}$ (4)

Using Eqs. 2 and 4, the shaft output power is estimated from the Euler turbomachine equation,

$$\begin{aligned}\dot{W}_{\text{shaft}} &= \rho\omega\dot{V}(r_2V_{2,t} - r_1V_{1,t}) = (998.0 \text{ kg/m}^3)(18.85 \text{ rad/s})(340 \text{ m}^3/\text{s})\times[(2.00 \text{ m})(21.4 \text{ m/s}) - (1.42 \text{ m})(3.05 \text{ m/s})] \\ &= 2.46\times10^8 \frac{\text{kg}\cdot\text{m}^2}{\text{s}^3}\left(\frac{\text{MW}\cdot\text{s}^3}{10^6 \text{ kg}\cdot\text{m}^2}\right) = 246.055 \text{ MW} \cong \mathbf{246 \text{ MW}}\end{aligned} \quad (5)$$

Finally, we calculate the required net head using Eq. 14-44, assuming that $\eta_{\text{turbine}}$ = 100% since we are ignoring irreversibilities,

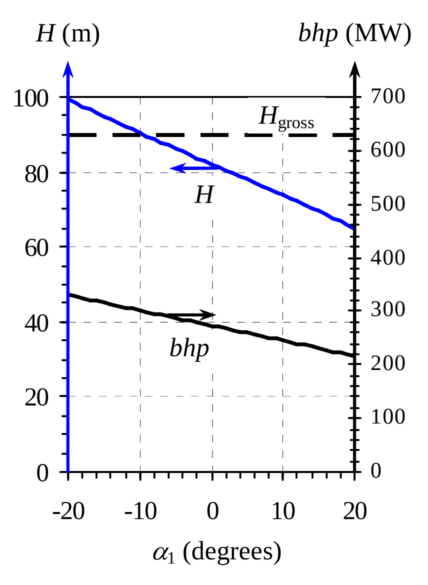
$$H = \frac{bhp}{\rho g\dot{V}} = \frac{246.055 \text{ MW}}{(998.0 \text{ kg/m}^3)(9.81 \text{ m/s}^2)(340 \text{ m}^3/\text{s})}\left(\frac{10^6 \text{ kg}\cdot\text{m}^2}{\text{MW}\cdot\text{s}^3}\right) = \mathbf{73.9 \text{ m}} \quad (6)$$

Since the required net head is less than the gross net head, **the design is feasible**.

***Discussion*** This is a preliminary design in which we are neglecting irreversibilities. Actual output power will be lower, and actual required net head will be higher than the values predicted here.

**14-79** [*Also solved using EES on enclosed DVD*]

**Solution** We are to examine the effect of runner outlet angle $\alpha_1$ on the required net head and the output power of a hydroturbine.




***Assumptions*** **1** The flow is steady. **2** The fluid is water at 20°C. **3** The blades are infinitesimally thin. **4** The flow is everywhere tangent to the runner blades. **5** We neglect irreversible losses through the turbine.

***Properties*** For water at 20°C, $\rho$ = 998.0 kg/m$^3$.

***Analysis*** We repeat the calculations of the previous problem, but with $\alpha_1$ varying from -20° to 20°. The results are plotted. We see that the required net head $H$ and the output brake horsepower *bhp* decrease as $\alpha_1$ increases. This agrees with our expectations, based on the negative sign on the $V_{1,t}$ term in the Euler turbomachine equation. In other words, we can produce greater power by increasing the reverse swirl, but at the cost of increased required net head. However, when $\alpha_1$ is smaller than about -9°, the required net head rises above $H_{gross}$, which is impossible, even with no irreversibilities. Thus, **when $\alpha_1$ < -9°, the predicted net head and brake horsepower are not feasible – they violate the second law of thermodynamics**.

***Discussion*** A small amount of reverse swirl is usually good, but too much reverse swirl is not good.

**14-80**

**Solution** We are to calculate flow angles, required net head, and power output for a Francis turbine.

***Assumptions*** **1** The flow is steady. **2** The fluid is water at 20°C. **3** The blades are infinitesimally thin. **4** The flow is everywhere tangent to the runner blades. **5** We neglect irreversible losses through the turbine.

***Properties*** For water at 20°C, $\rho$ = 998.0 kg/m³.

***Analysis*** The angular velocity is $\omega = 2\pi\dot{n}/60$ = 10.47 rad/s. We solve for the normal component of velocity at the inlet,

$$V_{2,n} = \frac{\dot{V}}{2\pi r_2 b_2} = \frac{80.0\ \text{m}^3/\text{s}}{2\pi(2.00\ \text{m})(0.85\ \text{m})} = 7.490\ \text{m/s} \qquad (1)$$

The tangential velocity component of the absolute velocity at the inlet is obtained from trigonometry to be (see Eq. 14-45)

$$V_{2,t} = \omega r_2 - \frac{V_{2,n}}{\tan\beta_2} = (10.47\ \text{rad/s})(2.00\ \text{m}) - \frac{7.490\ \text{m/s}}{\tan 66^\circ} = 17.61\ \text{m/s} \qquad (2)$$

From these two components of $V_2$ in the absolute coordinate system, we calculate the angle $\alpha_2$ through which the wicket gates should turn the flow,

$$\alpha_2 = \tan^{-1}\frac{V_{2,t}}{V_{2,n}} = \tan^{-1}\frac{17.61\ \text{m/s}}{7.490\ \text{m/s}} = 66.95^\circ \approx \mathbf{67^\circ} \qquad (3)$$

In exactly similar fashion, we solve for the velocity components and swirl angle at the runner outlet. We get

*Runner outlet*:

$$V_{1,n} = 4.664\ \text{m/s}, \quad V_{1,t} = -0.3253\ \text{m/s}, \quad \alpha_1 = -3.990^\circ \approx \mathbf{-4.0^\circ} \qquad (4)$$

Since $\alpha_1$ is negative, **this turbine operates with a small amount of reverse swirl**.

Using the tangential velocity components of Eqs. 2 and 4, the shaft output power is estimated from the Euler turbomachine equation,

$$\begin{aligned}\dot{W}_{\text{shaft}} &= \rho\omega\dot{V}\left(r_2V_{2,t} - r_1V_{1,t}\right) = (998.0\ \text{kg/m}^3)(10.47\ \text{rad/s})(80.0\ \text{m}^3/\text{s})\\ &\quad\times\left[(2.00\ \text{m})(17.61\ \text{m/s}) - (1.30\ \text{m})(-0.3253\ \text{m/s})\right]\\ &= 2.98\times10^7\ \frac{\text{kg}\cdot\text{m}^2}{\text{s}^3}\left(\frac{\text{MW}\cdot\text{s}^3}{10^6\ \text{kg}\cdot\text{m}^2}\right) = \mathbf{29.8\ MW}\end{aligned} \qquad (5)$$

Finally, we calculate the required net head using Eq. 14-44, assuming that $\eta_{\text{turbine}}$ = 100% since we are ignoring irreversibilities,

$$H = \frac{bhp}{\rho g\dot{V}} = \frac{29.8\ \text{MW}}{(998.0\ \text{kg/m}^3)(9.81\ \text{m/s}^2)(80.0\ \text{m}^3/\text{s})}\left(\frac{10^6\ \text{kg}\cdot\text{m}^2}{\text{MW}\cdot\text{s}^3}\right) = \mathbf{38.0\ m} \qquad (6)$$

***Discussion*** This is a preliminary design in which we are neglecting irreversibilities. Actual output power will be lower, and actual required net head will be higher than the values predicted here. Notice the double negative in the outlet terms of Eq. 5 – reverse swirl leads to greater performance, but requires more head.

**14-81E**

**Solution** We are to calculate flow angles, required net head, and power output for a Francis turbine.

***Assumptions*** **1** The flow is steady. **2** The fluid is water at 68°F. **3** The blades are infinitesimally thin. **4** The flow is everywhere tangent to the runner blades. **5** We neglect irreversible losses through the turbine.

***Properties*** For water at 68°F, $\rho$ = 62.32 lbm/ft$^3$.

***Analysis*** The angular velocity is $\omega = 2\pi\dot{n}/60$ = 12.57 rad/s. The volume flow rate is (4.70 × 10$^6$ gpm)[ft$^3$/s / (448.83 gpm)] = 10,470 ft$^3$/s. We solve for the normal component of velocity at the inlet,

$$V_{2,n} = \frac{\dot{V}}{2\pi r_2 b_2} = \frac{10,470\ \text{ft}^3/\text{s}}{2\pi(6.60\ \text{ft})(2.60\ \text{ft})} = 97.1\ \text{ft/s} \qquad (1)$$

The tangential velocity component of the absolute velocity at the inlet is obtained from trigonometry to be (see Eq. 14-45)

$$V_{2,t} = \omega r_2 - \frac{V_{2,n}}{\tan\beta_2} = (12.57\ \text{rad/s})(6.60\ \text{ft}) - \frac{97.12\ \text{ft/s}}{\tan 82^\circ} = 69.3\ \text{ft/s} \qquad (2)$$

From these two components of $V_2$ in the absolute coordinate system, we calculate the angle $\alpha_2$ through which the wicket gates should turn the flow,

$$\alpha_2 = \tan^{-1}\frac{V_{2,t}}{V_{2,n}} = \tan^{-1}\frac{69.3\ \text{ft/s}}{97.1\ \text{ft/s}} = \mathbf{35.5^\circ} \qquad (3)$$

In exactly similar fashion, we solve for the velocity components and swirl angle at the runner outlet. We get

*Runner outlet*: $$V_{1,n} = 52.6\ \text{ft/s}, \quad V_{1,t} = 4.49\ \text{ft/s}, \quad \alpha_1 = \mathbf{4.9^\circ} \qquad (4)$$

Since $\alpha_1$ is positive, **this turbine operates with a small amount of forward swirl**.

Using the tangential velocity components of Eqs. 2 and 4, the shaft output power is estimated from the Euler turbomachine equation,

$$\begin{aligned}\dot{W}_{\text{shaft}} &= \rho\omega\dot{V}(r_2V_{2,t} - r_1V_{1,t}) = (62.32\ \text{lbm/ft}^3)(12.57\ \text{rad/s})(10,470\ \text{ft}^3/\text{s}) \\ &\times\left[(6.60\ \text{ft})(69.3\ \text{ft/s}) - (4.40\ \text{ft})(4.49\ \text{ft/s})\right]\left(\frac{\text{lbf}\cdot\text{s}^2}{32.174\ \text{lbm}\cdot\text{ft}}\right) \\ &= 1.116\times10^8\ \frac{\text{ft}\cdot\text{lbf}}{\text{s}}\left(\frac{1.8182\times10^{-3}\ \text{hp}\cdot\text{s}}{\text{ft}\cdot\text{lbf}}\right) = \mathbf{2.03\times10^5\ hp}\end{aligned} \qquad (5)$$

Finally, we calculate the required net head using Eq. 14-44, assuming that $\eta_{\text{turbine}}$ = 100% since we are ignoring irreversibilities,

$$H = \frac{bhp}{\rho g\dot{V}} = \frac{1.116\times10^8\ \text{ft}\cdot\text{lbf/s}}{(62.32\ \text{lbm/ft}^3)(32.174\ \text{ft/s}^2)(10,470\ \text{ft}^3/\text{s})}\left(\frac{32.174\ \text{lbm}\cdot\text{ft}}{\text{lbf}\cdot\text{s}^2}\right) = \mathbf{171\ ft} \qquad (6)$$

***Discussion*** This is a preliminary design in which we are neglecting irreversibilities. Actual output power will be lower, and actual required net head will be higher than the values predicted here.

## 14-82E



**Solution** We are to calculate the runner blade trailing edge angle such that there is no swirl. At this value of $\beta_1$, we are to also calculate the shaft power.

***Assumptions*** **1** The flow is steady. **2** The fluid is water at 68°F. **3** The blades are infinitesimally thin. **4** The flow is everywhere tangent to the runner blades. **5** We neglect irreversible losses through the turbine.

***Properties*** For water at 68°F, $\rho$ = 62.32 lbm/ft$^3$.

***Analysis*** Using EES or other software, we adjust $\beta_1$ by trial and error until $\alpha_1$ = 0. It turns out that $\beta_1$ = **43.6°**, at which $\dot{W}_{\text{shaft}}$ = $\mathbf{2.12 \times 10^5}$ **hp**.

***Discussion*** It turns out that the swirl angle at the runner output is a strong function of $\beta_1$ – a small change in $\beta_1$ leads to a large change in $\alpha_1$. The shaft power increases, as expected, since the original swirl angle was positive. The increase in shaft power is less than 5%.

---

## Pump and Turbine Scaling Laws

## 14-83C

**Solution** We are to give a definition of "affinity", and explain why the scaling laws are called "affinity laws".

***Analysis*** Among the many definitions of "affinity" is "**inherent likeness or agreement**", and "...**resemblance in general plan or structure**". When two pumps or two turbines are geometrically similar and operate under dynamically similar conditions, they indeed have "inherent likeness". Thus, **the phrase "affinity laws" is appropriate for the scaling laws of turbomachinery**.

***Discussion*** Students will have various definitions, depending on the dictionary they use.

---

## 14-84C

**Solution**

(*a*) *True*: Rotation rate appears with an exponent of 1 in the affinity law for capacity. Thus, the change is linear.
(*b*) *False*: Rotation rate appears with an exponent of 2 in the affinity law for net head. Thus, if the rpm is doubled, the net head increases by a factor of 4.
(*c*) *False*: Rotation rate appears with an exponent of 3 in the affinity law for shaft power. Thus, if the rpm is doubled, the shaft power increases by a factor of 8.
(*d*) *True*: The affinity laws apply to turbines as well as pumps, so this statement is true, as discussed in Part (c).

---

## 14-85C

**Solution** We are to discuss which pump and turbine performance parameters are used as the independent parameter, and explain why

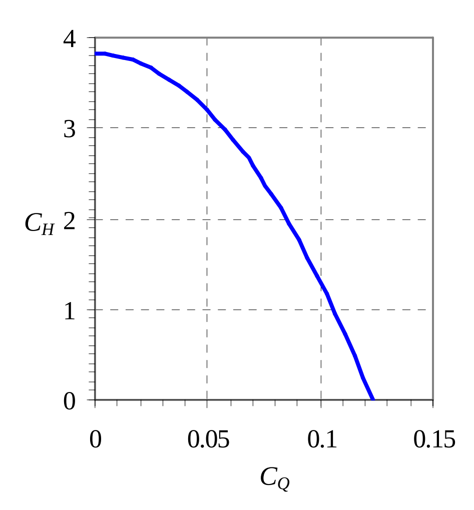
***Analysis*** For pumps, we use $C_Q$, the **capacity coefficient**, as the independent parameter. The reason is that the goal of a pump is to move fluid from one place to another, and the most important parameter is the pump's capacity (volume flow rate). On the other hand, for most turbines, we use $C_P$, the **power coefficient**, as the independent parameter. The reason is that the goal of a turbine is to rotate a shaft, and the most important parameter is the turbine's brake horsepower

***Discussion*** There are exceptions. For example, when analyzing a positive-displacement turbine used to measure volume flow rate, capacity is more important than output shaft power, so one might use $C_Q$, instead of $C_P$ as the independent parameter.

---

**14-86**

**Solution** We are to nondimensionalize a pump performance curve.




***Analysis*** The pump's net head is approximated by the parabolic expression

*Pump performance curve*: $H = H_0 - a\dot{V}^2$ (1)

where shutoff head $H_0$ = 24.4 m of water column, coefficient $a$ = 0.0678 m/Lpm$^2$, available pump head $H_{available}$ is in units of meters of water column, and capacity $\dot{V}$ is in units of liters per minute (Lpm). By definition, head coefficient $C_H = (gH)/(\omega^2 D^2)$, and capacity coefficient $C_Q = (\dot{V})/(\omega D^3)$. To convert, we must be careful with units. For example, we must convert the rotation rate from rpm to radians per second. The rotational speed is

$$\omega = 4200\frac{\text{rot}}{\text{min}}\left(\frac{2\pi\ \text{rad}}{\text{rot}}\right)\left(\frac{1\ \text{min}}{60\ \text{s}}\right) = 439.823\frac{\text{rad}}{\text{s}}$$

Sample calculations at 14.0 Lpm are shown below.

$$C_Q = \frac{\dot{V}}{\omega D^3} = \frac{14.0\ \text{L/min}}{(439.823\ \text{rad/s})(0.0180\ \text{m})^3}\left(\frac{1\ \text{m}^3}{1000\ \text{L}}\right)\left(\frac{1\ \text{min}}{60\ \text{s}}\right) = 0.90966 \cong 0.0910$$

At this flow rate (14.0 Lpm), the net head is obtained from Eq. 1,

$$H = 24.4\ \text{m} - \left(0.0678\ \text{m/Lpm}^2\right)\left(14.0\ \text{Lpm}\right)^2 = 11.111\ \text{m} \cong 11.1\ \text{m}$$

from which the head coefficient can be calculated,

$$C_H = \frac{gH}{\omega^2 D^2} = \frac{\left(9.81\ \text{m/s}^2\right)\left(11.111\ \text{m}\right)}{\left(439.823\ \text{rad/s}\right)^2\left(0.0180\ \text{m}\right)^2} = 1.7391 \cong 1.74$$

These calculations are repeated for a range of volume flow rates from 0 to $\dot{V}_{max}$. The nondimensionalized pump performance curve is plotted.

***Discussion*** Since radians is a dimensionless unit, the units of $C_H$ and $C_Q$ are unity. In other words, $C_H$ and $C_Q$ are nondimensional parameters, as they must be.

## 14-87

**Solution** We are to calculate the specific speed of a water pump, and determine what kind of pump it is.

***Analysis*** First, we use the values of $C_H$ and $C_Q$ calculated in the previous problem at the BEP to calculate the dimensionless form of $N_{Sp}$,

*Dimensionless pump specific speed*:
$$N_{Sp} = \frac{C_Q^{\ 1/2}}{C_H^{\ 3/4}} = \frac{0.090966^{1/2}}{1.7391^{3/4}} = 0.1992$$

Thus, $\boldsymbol{N_{Sp}}$ **= 0.199**, and is dimensionless. From the conversion given in the text, $\boldsymbol{N_{Sp,\mathrm{US}}}$ **= 0.1992 × 2,734 = 545**. Alternatively, we can use the original dimensional data to calculate $N_{Sp,\mathrm{US}}$,

*Dimensional pump specific speed in customary US units*:
$$N_{Sp,\mathrm{US}} = \frac{(\dot{n}, \mathrm{rpm})(\dot{V}, \mathrm{gpm})^{1/2}}{(H, \mathrm{ft})^{3/4}} = \frac{(4200\ \mathrm{rpm})\left(14.0\ \mathrm{Lpm}\left(\frac{0.2642\ \mathrm{gpm}}{\mathrm{Lpm}}\right)\right)^{1/2}}{\left(11.111\ \mathrm{m}\left(\frac{\mathrm{ft}}{0.3048\ \mathrm{m}}\right)\right)^{3/4}} = 545$$

From either the dimensionless or the dimensional pump specific speed, Fig. 14-73 shows that **this is definitely a centrifugal pump**.

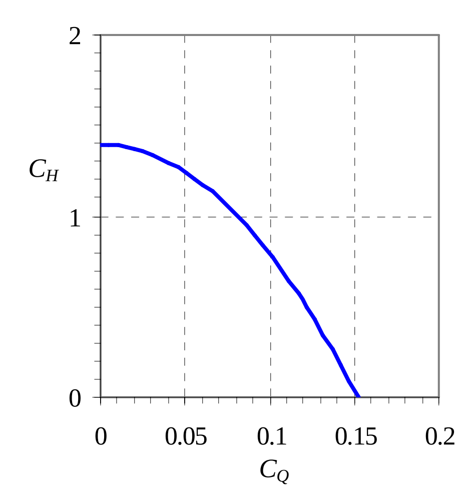
***Discussion*** We calculated the dimensional pump specific speed two ways as a check of our algebra, and (fortunately) the results agree.

---

**14-88**

**Solution** We are to nondimensionalize a fan performance curve.

***Analysis*** The fan's net head is approximated by the parabolic expression




*Fan performance curve*: $$H = H_0 - a\dot{V}^2 \quad (1)$$

where shutoff head $H_0$ = 60.0 mm of water column, coefficient $a = 2.50 \times 10^{-7}$ mm/Lpm$^2$, available fan head $H_{available}$ is in units of mm of water column, and capacity $\dot{V}$ is in units of liters per minute (Lpm). By definition, head coefficient $C_H = (gH)/(\omega^2 D^2)$, and capacity coefficient $C_Q = (\dot{V})/(\omega D^3)$. To convert, we must be careful with units. For example, we must convert the rotation rate from rpm to radians per second. The rotational speed becomes

$$\omega = 600\frac{\text{rot}}{\text{min}}\left(\frac{2\pi\ \text{rad}}{\text{rot}}\right)\left(\frac{1\ \text{min}}{60\ \text{s}}\right) = 62.8319\frac{\text{rad}}{\text{s}}$$

Sample calculations at 13,600 Lpm are shown below.

*Capacity coefficient at* 13,600 *Lpm*: $$C_Q = \frac{\dot{V}}{\omega D^3} = \frac{13,600\ \text{L/min}}{(62.8319\ \text{rad/s})(0.300\ \text{m})^3}\left(\frac{1\ \text{m}^3}{1000\ \text{L}}\right)\left(\frac{1\ \text{min}}{60\ \text{s}}\right) = 0.13361 \cong 0.134$$

At this flow rate, the net head is obtained from Eq. 1,

$$H = 60.0\ \text{mm} - \left(2.50\times10^{-7}\ \text{mm/Lpm}^2\right)(13,600\ \text{Lpm})^2 = 13.76\ \text{mm H}_2\text{O}$$

from which the head coefficient can be calculated,

*Head coefficient at* 13,600 *Lpm*: $$C_H = \frac{gH}{\omega^2 D^2} = \frac{(9.81\ \text{m/s}^2)(0.01376\ \text{m H}_2\text{O})}{(62.8319\ \text{rad/s})^2(0.300\ \text{m})^2}\frac{998.\ \text{kg/m}^3\ (\text{m air})}{1.184\ \text{kg/m}^3\ (\text{m H}_2\text{O})} = 0.32023 \cong 0.320$$

Note the ratio of water density to air density to convert the head from water column height to air column height. The calculations are repeated for a range of volume flow rates from 0 to $\dot{V}_{max}$. The nondimensionalized pump performance curve is plotted.

***Discussion*** Since radians is a dimensionless unit, the units of $C_H$ and $C_Q$ are unity. In other words, $C_H$ and $C_Q$ are nondimensional parameters, as they must be.

**14-89**

**Solution** We are to calculate the specific speed of a fan, and determine what kind of fan it is.

***Analysis*** First, we use the values of $C_H$ and $C_Q$ calculated in the previous problem at the BEP (to four significant digits) to calculate the dimensionless form of $N_{Sp}$,

*Dimensionless pump specific speed*:

$$N_{Sp} = \frac{C_Q^{\ 1/2}}{C_H^{\ 3/4}} = \frac{0.13361^{1/2}}{0.32023^{3/4}} = 0.8587$$

Thus, $\boldsymbol{N_{Sp}}$ **= 0.859**, and is dimensionless. From the conversion given in the text, $\boldsymbol{N_{Sp,\mathrm{US}}}$ **= 0.8587 × 2,734 = 2350**. Alternatively, we can use the dimensional data to calculate $N_{Sp,\mathrm{US}}$, the dimensional pump specific speed in US units,

$$N_{Sp,\mathrm{US}} = \frac{(\dot{n},\ \mathrm{rpm})(\dot{V},\ \mathrm{gpm})^{1/2}}{(H,\ \mathrm{ft})^{3/4}} = \frac{(600\ \mathrm{rpm})\left(13,600\ \mathrm{Lpm}\left(\frac{0.2642\ \mathrm{gpm}}{\mathrm{Lpm}}\right)\right)^{1/2}}{\left(11.60\ \mathrm{m}\left(\frac{\mathrm{ft}}{0.3048\ \mathrm{m}}\right)\right)^{3/4}} = 2350$$

Note that we use $H$ = 11.60 m *of air*, since air is the fluid being pumped here. We calculate $H$ as $H_{\mathrm{air}} = H_{\mathrm{water}}(\rho_{\mathrm{water}}/\rho_{\mathrm{air}})$ = (0.01376 m of water)(998/1.184) = 11.60 m of air. From either the dimensionless or the dimensional pump specific speed, Fig. 14-73 shows that **this is most likely a centrifugal fan** (probably a squirrel cage fan).

***Discussion*** We calculated the dimensional pump specific speed two ways as a check of our algebra, and (fortunately) the results agree.

---

**14-90**

**Solution** We are to calculate the specific speed of a water pump, and determine what kind of pump it is.

***Analysis*** First, we use the values of $C_H$ and $C_Q$ calculated in Example 14-11 at the BEP to calculate the dimensionless form of $N_{Sp}$,

*Dimensionless pump specific speed*:

$$N_{Sp} = \frac{C_Q^{\ 1/2}}{C_H^{\ 3/4}} = \frac{0.0112^{1/2}}{0.133^{3/4}} = 0.481$$

Thus, $\boldsymbol{N_{Sp}}$ **= 0.481**. From the conversion given in the text, $\boldsymbol{N_{Sp,\mathrm{US}}}$ **= 1310**. From Fig. 14-73, **this is most likely a centrifugal pump**.

***Discussion*** From Fig. 14-73, the maximum efficiency one can expect from a centrifugal pump at this value of $N_{Sp}$ is about 0.87 (87%). The actual pump efficiency is only 81.2% (from Example 14-11), so there is room for improvement in the design.

---

**14-91**

**Solution** We are to decide what king of pump should be designed for given performance criteria.

***Analysis*** We calculate the pump specific speed at the given conditions,

$$N_{Sp} = \frac{\omega\dot{V}^{1/2}}{(gH)^{3/4}} = \frac{(1200\ \mathrm{rpm})\left(\frac{2\pi}{60}\frac{\mathrm{rad/s}}{\mathrm{rpm}}\right)\left(18.0\ \mathrm{L/min}\left(\frac{1\ \mathrm{m}^3}{1000\ \mathrm{L}}\right)\left(\frac{1\ \mathrm{min}}{60\ \mathrm{s}}\right)\right)^{1/2}}{\left((9.81\ \mathrm{m/s}^2)(1.6\ \mathrm{m})\right)^{3/4}} = 0.276$$

From Fig. 14-73, we see that **Len should design a centrifugal pump**. **The maximum pump efficiency is about 0.75 (75%)** (based on Fig. 14-73 again).

***Discussion*** This value of $N_{Sp}$ is, in fact, on the low end of the curve for centrifugal pumps, so a centrifugal pump is the best Len can do, in spite of the low efficiency.

---

**14-92E**

**Solution** We are to decide what king of pump should be designed for given performance criteria.

***Properties*** The density of water at 77°F is 62.24 lbm/ft$^3$.

***Analysis*** We calculate the pump specific speed at the given conditions,

$$N_{Sp} = \frac{\omega \dot{V}^{1/2}}{(gH)^{3/4}} = \frac{(300\text{ rpm})\left(\frac{2\pi}{60}\frac{\text{rad/s}}{\text{rpm}}\right)\left(2{,}500\text{ gal/min}\left(\frac{0.13368\text{ ft}^3}{1\text{ gal}}\right)\left(\frac{1\text{ min}}{60\text{ s}}\right)\right)^{1/2}}{\left((32.174\text{ ft/s}^2)(45\text{ ft})\right)^{3/4}} = 0.316$$

From Fig. 14-73, we choose **a centrifugal pump**. **The maximum pump efficiency is about 0.82 (82%)** (based on Fig. 14-73 again). From the definition of pump efficiency, $bhp = \eta_{\text{pump}}\rho\dot{V}gH$. Thus, the required brake horsepower is

$$bhp = \frac{\rho g H \dot{V}}{\eta_{\text{pump}}} = \frac{(62.24\text{ lbm/ft}^3)(32.174\text{ ft/s}^2)(45\text{ ft})(5.57\text{ ft}^3/\text{s})}{0.82}\left(\frac{\text{lbf}\cdot\text{s}^2}{32.174\text{ lbm}\cdot\text{ft}}\right) = 19{,}025\text{ ft}\cdot\text{lbf/s}\left(\frac{1\text{ hp}}{550\text{ ft}\cdot\text{lbf/s}}\right) = 34.6\text{ hp}$$

Thus, we expect that **the pump will require about 35 hp to turn the shaft**.

***Discussion*** Most large water pumps are of the centrifugal variety. This problem may also be solved in terms of the dimensional pump specific speed in customary US units: $N_{Sp,\text{US}} = 864$.

---

**14-93**

**Solution** We are to calculate the performance of a modified pump.

***Assumptions*** **1** The modified pump operates at the best efficiency point. **2** Pump diameter and fluid properties remain the same.

***Analysis*** At homologous points, the affinity laws are used to estimate the operating conditions of the modified pump. We let the original pump be Pump A, and the modified pump be Pump B. We get

*Volume flow rate*:
$$\dot{V}_B = \dot{V}_A \frac{\omega_B}{\omega_A}\underbrace{\left(\frac{D_B}{D_A}\right)^3}_{1} = (18.\text{ L/min})\frac{600\text{ rpm}}{1200\text{ rpm}} = 9.0\text{ L/min}$$

and

*Net head*:
$$H_B = H_A\left(\frac{\omega_B}{\omega_A}\right)^2\underbrace{\left(\frac{D_B}{D_A}\right)^2}_{1} = (1.6\text{ m})\left(\frac{600\text{ rpm}}{1200\text{ rpm}}\right)^2 = 0.40\text{ m}$$

**The volume flow rate of the modified pump is 9.0 L/min; the net head is 0.40 m**. The pump specific speed of the modified pump is

$$N_{Sp} = \frac{\omega \dot{V}^{1/2}}{(gH)^{3/4}} = \frac{(600\text{ rpm})\left(\frac{2\pi}{60}\frac{\text{rad/s}}{\text{rpm}}\right)\left(9.0\text{ L/min}\left(\frac{1\text{ m}^3}{1000\text{ L}}\right)\left(\frac{1\text{ min}}{60\text{ s}}\right)\right)^{1/2}}{\left((9.81\text{ m/s}^2)(0.40\text{ m})\right)^{3/4}} = 0.276$$

Thus, $\boldsymbol{N_{Sp}}$ **= 0.276**, which is the same as that of the original pump. This is not surprising since the two pumps operate at homologous points.

***Discussion*** When the rpm is cut in half, all else being equal, the volume flow rate of the pump decreases by a factor of two, while the net head decreases by a factor of four. This agrees with the mnemonic of Fig. 14-74. The specific speed of the two pumps must match since they operate at homologous points.

---

**14-94**

**Solution** We are to prove the relationship between turbine specific speed and pump specific speed.

***Assumptions*** **1** The parameters such as net head, volume flow rate, diameter, etc. are defined in similar fashion for the pump and the turbine.

***Analysis*** First we write the definitions of pump specific speed and turbine specific speed,

*Pump specific speed*:
$$N_{Sp} = \frac{\omega \dot{V}^{1/2}}{(gH)^{3/4}} \qquad (1)$$

and

*Turbine specific speed*:
$$N_{St} = \frac{\omega (bhp)^{1/2}}{(\rho)^{1/2} (gH)^{5/4}} \qquad (2)$$

After some rearrangement of Eq. 2,

$$N_{St} = \frac{\omega \dot{V}^{1/2}}{(gH)^{3/4}} \left( \frac{bhp}{\rho g H \dot{V}} \right)^{1/2} \qquad (3)$$

We recognize the first grouping of terms in Eq. 3 as $N_{Sp}$ and the second grouping of terms as the square root of turbine efficiency $\eta_{\text{turbine}}$. Thus,

*Final relationship*:
$$\boxed{N_{St} = N_{Sp} \sqrt{\eta_{\text{turbine}}}} \qquad (4)$$

***Discussion*** Since turbine efficiency is typically large (90 to 95% for large hydroturbines), pump specific speed and turbine specific speed are nearly equivalent. Note that Eq. 4 does *not* apply to a pump running backwards as a turbine or vice-versa. Such devices, called *pump-turbines*, are addressed in the next problem.

**14-95**

**Solution** We are to prove the relationship between turbine specific speed and pump specific speed for the case of a pump-turbine operating at the same volume flow rate and rotational speed when acting as a pump and as a turbine.

***Assumptions*** **1** The parameters such as net head, volume flow rate, diameter, etc. are defined in similar fashion for the pump and the turbine.

***Analysis*** First we write the definitions of pump specific speed and turbine specific speed,

*Pump specific speed*:
$$N_{Sp} = \frac{\omega \dot{V}^{1/2}}{\left(gH_{\text{pump}}\right)^{3/4}} \qquad (1)$$

and

*Turbine specific speed*:
$$N_{St} = \frac{\omega \left(bhp_{\text{turbine}}\right)^{1/2}}{\left(\rho\right)^{1/2}\left(gH_{\text{turbine}}\right)^{5/4}} \qquad (2)$$

Note that we have added subscripts "pump" and "turbine" on net head and brake horsepower since $H_{\text{pump}} \neq H_{\text{turbine}}$ and $bhp_{\text{pump}} \neq bhp_{\text{turbine}}$. After some rearrangement of Eq. 2,

$$N_{St} = \frac{\omega \dot{V}^{1/2}}{\left(gH_{\text{turbine}}\right)^{3/4}}\left(\frac{bhp_{\text{turbine}}}{\rho g H_{\text{turbine}}\dot{V}}\right)^{1/2} = \frac{\omega \dot{V}^{1/2}}{\left(gH_{\text{turbine}}\right)^{3/4}}\left(\eta_{\text{turbine}}\right)^{1/2} \qquad (3)$$

We also write the definitions of pump efficiency and turbine efficiency,

$$\eta_{\text{pump}} = \frac{\rho g \dot{V} H_{\text{pump}}}{bhp_{\text{pump}}} \qquad \eta_{\text{turbine}} = \frac{bhp_{\text{turbine}}}{\rho g \dot{V} H_{\text{turbine}}} \qquad (4)$$

We solve both parts of Eq. 4 for $\dot{V}$ and equate the two, since $\dot{V}$ is the same whether the pump-turbine is operating as a pump or as a turbine. Eliminating $\dot{V}$,

$$H_{\text{turbine}} = \frac{H_{\text{pump}} bhp_{\text{turbine}}}{bhp_{\text{pump}}\eta_{\text{pump}}\eta_{\text{turbine}}} \qquad (5)$$

where $\rho$ and $g$ have also dropped out since they are the same for both cases. We plug Eq. 5 into Eq. 3 and rearrange,

$$N_{St} = \frac{\omega \dot{V}^{1/2}}{\left(gH_{\text{pump}}\right)^{3/4}}\left(\eta_{\text{turbine}}\right)^{1/2}\left(\frac{bhp_{\text{pump}}\eta_{\text{pump}}\eta_{\text{turbine}}}{bhp_{\text{turbine}}}\right)^{3/4} \qquad (6)$$

We recognize the first grouping of terms in Eq. 6 as $N_{Sp}$ and rearrange,

*Final relationship*:
$$\boxed{N_{St} = N_{Sp}\left(\eta_{\text{turbine}}\right)^{5/4}\left(\eta_{\text{pump}}\right)^{3/4}\left(\frac{bhp_{\text{pump}}}{bhp_{\text{turbine}}}\right)^{3/4}} \qquad (7)$$

An alternate expression is obtained in terms of net heads by substitution of Eq. 5,

*Alternate final relationship*:
$$\boxed{N_{St} = N_{Sp}\sqrt{\eta_{\text{turbine}}}\left(\frac{H_{\text{pump}}}{H_{\text{turbine}}}\right)^{3/4}} \qquad (8)$$

***Discussion*** It is difficult to achieve high efficiency in a pump-turbine during both the pump duty cycle *and* the turbine duty cycle. To achieve the highest possible efficiency, it is critical that both $N_{Sp}$ and $N_{St}$ are appropriate for the chosen design, e.g. radial flow centrifugal pump and radial-flow Francis turbine.

**14-96**

**Solution** We are to apply conversion factors to prove a conversion factor.

***Properties*** We set $g$ = 32.174 ft/s$^2$ and assume water at density $\rho$ = 1.94 slug/ft$^3$.

***Analysis*** We convert $N_{St,\text{US}}$ to $N_{St}$ by dividing by $g^{5/4}$ and $\rho^{1/2}$, and then using conversion ratios to cancel all units,

$$N_{St} = \underbrace{\frac{(\omega, \text{rot/min})(bhp, \text{hp})^{1/2}}{(H, \text{ft})^{5/4}}}_{N_{St,\text{US}}} \frac{1}{(1.94 \text{ slug/ft}^3)^{1/2}(32.174 \text{ ft/s}^2)^{5/4}} \times \left(\frac{1 \text{ min}}{60 \text{ s}}\right)\left(\frac{\text{slug ft}}{\text{s}^2 \text{ lbf}}\right)^{1/2}\left(\frac{550 \text{ ft lbf}}{\text{s hp}}\right)^{1/2}\left(\frac{2\pi \text{ rad}}{\text{s}}\right)$$

$$N_{St} = 0.02301 N_{St,\text{US}}$$

Finally, the inverse of the above equation yields the desired conversion factor.

***Discussion*** As discussed in the text, some turbomachinery authors do not convert rotations to radians, introducing a confusing factor of $2\pi$ into the conversion.

---

**14-97**

**Solution** We are to calculate turbine specific speed for a given hydroturbine and compare to the guidelines of Fig. 14-108.

***Assumptions*** **1** The fluid is water at 20°C.

***Properties*** The density of water at 20°C is $\rho$ = 998.0 kg/m$^3$.

***Analysis*** First we convert the turbine rotation rate from rpm to radians/s,

*Rotational speed*: $$\omega = 180 \frac{\text{rot}}{\text{min}}\left(\frac{2\pi \text{ rad}}{\text{rot}}\right)\left(\frac{1 \text{ min}}{60 \text{ s}}\right) = 18.85 \text{ rad/s}$$

With all other parameters already given in SI units, we calculate turbine specific speed,

$$N_{St} = \frac{\omega (bhp)^{1/2}}{(\rho)^{1/2}(gH)^{5/4}} = \frac{(18.85 \text{ rad/s})(119\times10^6 \text{ W})^{1/2}}{(998 \text{ kg/m}^3)^{1/2}\left[(9.81 \text{ m/s}^2)(105 \text{ m})\right]^{5/4}}\left(\frac{\text{m}^2\text{kg}}{\text{s}^3 \cdot \text{W}}\right)^{1/2} = \mathbf{1.12}$$

Comparing to the turbine specific speed guidelines of Fig. 14-108, we see that this turbine should be of the Francis type, which it is.

***Discussion*** In fact, the turbine specific speed of this turbine is close to that which yields the maximum possible efficiency for a Francis turbine.

---

## 14-98

**Solution** We are to calculate turbine specific speed for a given hydroturbine and compare to the guidelines of Fig. 14-108.

***Assumptions*** **1** The fluid is water at 20°C.

***Properties*** The density of water at 20°C is $\rho$ = 998.0 kg/m³.

***Analysis*** First we convert the turbine rotation rate from rpm to radians/s,

*Rotational speed*:
$$\omega = 100\frac{\text{rot}}{\text{min}}\left(\frac{2\pi\ \text{rad}}{\text{rot}}\right)\left(\frac{1\ \text{min}}{60\ \text{s}}\right) = 10.47\ \text{rad/s}$$

With all other parameters already given in SI units, we calculate turbine specific speed,

$$N_{St} = \frac{\omega(bhp)^{1/2}}{(\rho)^{1/2}(gH)^{5/4}} = \frac{(10.47\ \text{rad/s})(194\times10^6\ \text{W})^{1/2}}{(998\ \text{kg/m}^3)^{1/2}\left[(9.81\ \text{m/s}^2)(54.9\ \text{m})\right]^{5/4}}\left(\frac{\text{m}^2\text{kg}}{\text{s}^3\cdot\text{W}}\right)^{1/2} = \mathbf{1.78}$$

Comparing to the turbine specific speed guidelines of Fig. 14-108, we see that this turbine should be of the Francis type, which it is.

***Discussion*** The turbine specific speed of this turbine is close to the crossover point between a Francis turbine and a Kaplan turbine.

---

## 14-99

**Solution** We are to calculate turbine specific speed for a given hydroturbine and compare to the guidelines of Fig. 14-108.

***Assumptions*** **1** The fluid is water at 20°C.

***Properties*** The density of water at 20°C is $\rho$ = 998.0 kg/m³.

***Analysis*** First we convert the turbine rotation rate from rpm to radians/s,

*Rotational speed*:
$$\omega = 100\frac{\text{rot}}{\text{min}}\left(\frac{2\pi\ \text{rad}}{\text{rot}}\right)\left(\frac{1\ \text{min}}{60\ \text{s}}\right) = 10.47\ \text{rad/s}$$

With all other parameters already given in SI units, we calculate turbine specific speed,

$$N_{St} = \frac{\omega(bhp)^{1/2}}{(\rho)^{1/2}(gH)^{5/4}} = \frac{(10.47\ \text{rad/s})(5.37\times10^6\ \text{W})^{1/2}}{(998\ \text{kg/m}^3)^{1/2}\left[(9.81\ \text{m/s}^2)(9.75\ \text{m})\right]^{5/4}}\left(\frac{\text{m}^2\text{kg}}{\text{s}^3\cdot\text{W}}\right)^{1/2} = \mathbf{2.57}$$

Comparing to the turbine specific speed guidelines of Fig. 14-108, we see that this turbine should be of the Kaplan type, which it is.

***Discussion*** In fact, the turbine specific speed of this turbine is close to that which yields the maximum possible efficiency for a Kaplan turbine.

---

**14-100**

**Solution** We are to calculate the specific speed of a turbine, and compare it to the normal range.

***Analysis*** We first calculate the nondimensional form of $N_{St}$,

$$N_{St} = \frac{\omega (bhp)^{1/2}}{\rho^{1/2} (gH)^{5/4}} = \frac{(12.57 \text{ rad/s})(4.61\times 10^8 \text{ W})^{1/2}}{(998.0 \text{ kg/m}^3)^{1/2}\left[(9.81 \text{ m/s}^2)(78.6 \text{ m})\right]^{5/4}}\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right)^{1/2} = \mathbf{2.10}$$

From Fig. 14-108, this is on the high end for Francis turbines – the designers may wish to consider a Kaplan turbine instead. From the conversion given in the text, $\boldsymbol{N_{St,\text{US}}}$ **= 2.10 × 43.46 = 91.4**. Alternatively, we can use the original dimensional data to calculate $N_{St,\text{US}}$,

$$N_{St,\text{US}} = \frac{(\dot{n}, \text{rpm})(bhp, \text{hp})^{1/2}}{(H, \text{ft})^{5/4}} = \frac{(120 \text{ rpm})(6.18\times 10^5 \text{ hp})^{1/2}}{\left(78.6 \text{ m}\left(\frac{\text{ft}}{0.3048 \text{ m}}\right)\right)^{5/4}} = \mathbf{91.3}$$

Note that the actual values of brake horsepower and net head will differ from the values calculated here, because we have neglected irreversible losses; thus, the values of $N_{St}$ and $N_{St,\text{US}}$ may change slightly.

***Discussion*** We calculated the dimensional pump specific speed two ways as a check of our algebra, and (fortunately) the results agree (within round-off error).

---

**14-101**

**Solution** We are to calculate the specific speed of a turbine, and compare it to the normal range.

***Analysis*** We first calculate the nondimensional form of $N_{St}$,

$$N_{St} = \frac{\omega (bhp)^{1/2}}{\rho^{1/2} (gH)^{5/4}} = \frac{(10.47 \text{ rad/s})(2.98\times 10^7 \text{ W})^{1/2}}{(998.0 \text{ kg/m}^3)^{1/2}\left[(9.81 \text{ m/s}^2)(38.05 \text{ m})\right]^{5/4}}\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right)^{1/2} = \mathbf{1.10}$$

From Fig. 14-108, **this is in the range of the typical Francis turbine**. From the conversion given in the text, $\boldsymbol{N_{St,\text{US}}}$ **= 1.10 × 43.46 = 47.9**. Alternatively, we can use the original dimensional data to calculate $N_{St,\text{US}}$,

$$N_{St,\text{US}} = \frac{(\dot{n}, \text{rpm})(bhp, \text{hp})^{1/2}}{(H, \text{ft})^{5/4}} = \frac{(100 \text{ rpm})(4.00\times 10^4 \text{ hp})^{1/2}}{(124.8 \text{ ft})^{5/4}} = \mathbf{47.9}$$

Note that the actual values of brake horsepower and net head will differ from the values calculated here, because we have neglected irreversible losses; thus, the values of $N_{St}$ and $N_{St,\text{US}}$ may change slightly.

***Discussion*** We calculated the dimensional pump specific speed two ways as a check of our algebra, and the results agree.

---

## 14-102E

**Solution** We are to calculate the specific speed of a turbine, and compare it to the normal range.

***Analysis*** We first calculate the specific speed in customary US units,

$$N_{St,\mathrm{US}} = \frac{(\dot{n},\ \mathrm{rpm})(bhp,\ \mathrm{hp})^{1/2}}{(H,\ \mathrm{ft})^{5/4}} = \frac{(120\ \mathrm{rpm})(202{,}700\ \mathrm{hp})^{1/2}}{(170.8\ \mathrm{ft})^{5/4}} = \mathbf{87.5}$$

From Fig. 14-108, **this is on the high end of typical Francis turbines, on the border between Francis and Kaplan turbines**. In nondimensional terms, $\boldsymbol{N_{St,\mathrm{US}}}$ **= 87.5 / 43.46 = 2.01**. Note that the actual values of brake horsepower and net head will differ from the values calculated here, because we have neglected irreversible losses; thus, the values of $N_{St}$ and $N_{St,\mathrm{US}}$ may change slightly.

***Discussion*** The actual values of brake horsepower and net head will differ from the values calculated here, because we have neglected irreversible losses; thus, the values of $N_{St}$ and $N_{St,\mathrm{US}}$ may change slightly.

---

## 14-103

**Solution** We are to calculate the specific speed of a turbine, and compare it to the normal range.

***Analysis*** We first calculate the nondimensional form of $N_{St}$,

$$N_{St} = \frac{\omega(bhp)^{1/2}}{\rho^{1/2}(gH)^{5/4}} = \frac{(18.85\ \mathrm{rad/s})(2.46\times10^{8}\ \mathrm{W})^{1/2}}{(998.0\ \mathrm{kg/m^3})^{1/2}\left[(9.81\ \mathrm{m/s^2})(73.8\ \mathrm{m})\right]^{5/4}}\left(\frac{\mathrm{kg\cdot m^2}}{\mathrm{W\cdot s^3}}\right)^{1/2} = \mathbf{2.49}$$

From Fig. 14-108, this is much higher than the typical Francis turbine – **the designers should consider a Kaplan turbine instead**. From the conversion given in the text, $\boldsymbol{N_{St,\mathrm{US}}}$ **= 2.49 × 43.46 = 108**. Alternatively, we can use the original dimensional data to calculate $N_{St,\mathrm{US}}$,

$$N_{St,\mathrm{US}} = \frac{(\dot{n},\ \mathrm{rpm})(bhp,\ \mathrm{hp})^{1/2}}{(H,\ \mathrm{ft})^{5/4}} = \frac{(180\ \mathrm{rpm})(3.29\times10^{5}\ \mathrm{hp})^{1/2}}{\left(73.8\ \mathrm{m}\left(\frac{\mathrm{ft}}{0.3048\ \mathrm{m}}\right)\right)^{5/4}} = \mathbf{108}$$

Note that the actual values of brake horsepower and net head will differ from the values calculated here, because we have neglected irreversible losses; thus, the values of $N_{St}$ and $N_{St,\mathrm{US}}$ may change slightly.

***Discussion*** We calculated the dimensional pump specific speed two ways as a check of our algebra, and the results agree.

---

## 14-104

**Solution** We are to determine a turbine's efficiency and what kind of turbine is being tested.

***Properties*** The density of water at $T$ = 20°C is 998.0 kg/m³.

***Analysis*** The turbine's efficiency is calculated first,

$$\eta_{\text{turbine}} = \frac{bhp}{\rho g H \dot{V}} = \frac{450\text{ W}}{\left(998.0\text{ kg/m}^3\right)\left(9.81\text{ m/s}^2\right)\left(15.0\text{ m}\right)\left(17.0\text{ m}^3\text{/hr}\right)}\left(\frac{3600\text{ s}}{\text{hr}}\right)\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right) = \mathbf{64.9\%}$$

After converting 1500 rpm to 157.1 rad/s, we calculate the nondimensional form of turbine specific speed,

$$N_{St} = \frac{\omega\left(bhp\right)^{1/2}}{\rho^{1/2}\left(gH\right)^{5/4}} = \frac{\left(157.1\text{ rad/s}\right)\left(450\text{ W}\right)^{1/2}}{\left(998.0\text{ kg/m}^3\right)^{1/2}\left[\left(9.81\text{ m/s}^2\right)\left(15.0\text{ m}\right)\right]^{5/4}}\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right)^{1/2} = \mathbf{0.206}$$

which we convert to customary US units, $\mathbf{N_{St,US} = 0.206 \times 43.46 = 8.94}$. This is most likely an **impulse turbine** (e.g., a Pelton wheel turbine).

***Discussion*** We could instead have used the dimensional units to directly calculate the turbine specific speed in customary US units. The result would be the same.

---

## 14-105

**Solution** We are to scale up the model turbine tests to the prototype turbine.

***Assumptions*** **1** The prototype and model are geometrically similar. **2** The tests are conducted under conditions of dynamic similarity. **3** The water is at the same temperature for both the model and the prototype (20°C).

***Properties*** The density of water at $T$ = 20°C is 998.0 kg/m³.

***Analysis*** We use the turbine scaling laws, starting with the head coefficient, and letting the model be turbine A and the prototype be turbine B,

$$C_{H,\text{A}} = \frac{gH_\text{A}}{\omega_\text{A}^2 D_\text{A}^2} = \frac{gH_\text{B}}{\omega_\text{B}^2 D_\text{B}^2} = C_{H,\text{B}} \quad\rightarrow\quad \omega_\text{B} = \omega_\text{A}\frac{D_\text{A}}{D_\text{B}}\sqrt{\frac{H_\text{B}}{H_\text{A}}} \quad\rightarrow\quad \omega_\text{B} = \left(1500\text{ rpm}\right)\frac{1}{5}\sqrt{\frac{50\text{ m}}{15.0\text{ m}}} = \mathbf{548\text{ rpm}}$$

We then use the capacity coefficient to calculate the volume flow rate of the prototype,

$$C_{Q,\text{A}} = \frac{\dot{V}_\text{A}}{\omega_\text{A} D_\text{A}^3} = \frac{\dot{V}_\text{B}}{\omega_\text{B} D_\text{B}^3} = C_{Q,\text{B}} \quad\rightarrow\quad \dot{V}_\text{B} = \dot{V}_\text{A}\frac{\omega_\text{B}}{\omega_\text{A}}\left(\frac{D_\text{B}}{D_\text{A}}\right)^3 \quad\rightarrow\quad \dot{V}_\text{B} = \left(17.0\text{ m}^3\text{/hr}\right)\frac{548\text{ rpm}}{1500\text{ rpm}}\left(\frac{5}{1}\right)^3 = \mathbf{776\text{ m}^3\text{/hr}}$$

Finally, we use the power coefficient to calculate the brake horsepower of the prototype,

$$C_{P,\text{A}} = \frac{bhp_\text{A}}{\omega_\text{A}^3 D_\text{A}^5} = \frac{bhp_\text{B}}{\omega_\text{B}^3 D_\text{B}^5} = C_{P,\text{B}} \quad\rightarrow\quad bhp_\text{B} = bhp_\text{A}\left(\frac{\omega_\text{B}}{\omega_\text{A}}\right)^3\left(\frac{D_\text{B}}{D_\text{A}}\right)^5 \quad\rightarrow\quad \dot{V}_\text{B} = \left(450\text{ W}\right)\left(\frac{548\text{ rpm}}{1500\text{ rpm}}\right)^3\left(\frac{5}{1}\right)^5 = \mathbf{68,500\text{ W}}$$

***Discussion*** All results are given to 3 significant digits, but we kept several extra digits in the intermediate calculations to reduce round-off errors.

---

## 14-106

**Solution** We are to compare the model and prototype efficiency and turbine specific speed to prove that they operate at homologous points.

***Properties*** The density of water at $T$ = 20°C is 998.0 kg/m³.

***Analysis*** The model turbine's efficiency was calculated in Problem 14-104 as 64.9%. We calculate the prototype turbine's efficiency as

$$\eta_{\text{turbine}} = \frac{bhp}{\rho g H \dot{V}} = \frac{68,500 \text{ W}}{\left(998.0 \text{ kg/m}^3\right)\left(9.81 \text{ m/s}^2\right)\left(50 \text{ m}\right)\left(776 \text{ m}^3\text{/hr}\right)}\left(\frac{3600 \text{ s}}{\text{hr}}\right)\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right) = \mathbf{64.9\%}$$

The efficiency of the prototype is the same as that of the model. Similarly, the turbine specific speed of the model turbine was calculated previously as 0.206. After converting 548 rpm to 57.36 rad/s, we calculate the nondimensional form of turbine specific speed for the prototype turbine,

$$N_{St} = \frac{\omega (bhp)^{1/2}}{\rho^{1/2}(gH)^{5/4}} = \frac{\left(57.36 \text{ rad/s}\right)\left(68,500 \text{ W}\right)^{1/2}}{\left(998.0 \text{ kg/m}^3\right)^{1/2}\left[\left(9.81 \text{ m/s}^2\right)\left(50 \text{ m}\right)\right]^{5/4}}\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right)^{1/2} = \mathbf{0.206}$$

Which is also the same as the previous calculation. Comparing to the results of Problem 14-104, we see that both $\eta_{\text{turbine}}$ and $N_{st}$ match between the model and the prototype. Thus, **the model and the prototype operate at homologous points**.

***Discussion*** Other nondimensional turbine parameters, like head coefficient, capacity coefficient, and power coefficient also match between model and prototype. We could instead have used dimensional units to calculate the turbine specific speed in customary US units.

---

## 14-107

**Solution** We are to estimate the actual efficiency of the prototype, and explain why it is higher than the model efficiency.

***Analysis*** We apply the Moody efficiency correction equation,

$$\eta_{\text{turbine, prototype}} \approx 1-\left(1-\eta_{\text{turbine, model}}\right)\left(\frac{D_{\text{model}}}{D_{\text{prototype}}}\right)^{1/5} = 1-\left(1-0.649\right)\left(\frac{1}{5}\right)^{1/5} = \mathbf{74.6\%}$$

This represents an increase of 74.6 – 64.9 = 9.7%. However, as mentioned in the text, we expect only about 2/3 of this increase, or 2(9.7%)/3 = 6.5%. Thus, our best estimate of the actual efficiency of the prototype is

$$\eta_{\text{turbine, prototype}} \approx 64.9 + 6.5 = \mathbf{71.4\%}$$

There are several reasons for this increase in efficiency: The prototype turbine often operates at high Reynolds numbers that are not achievable in the laboratory. We know from the Moody chart that friction factor decreases with Re, as does boundary layer thickness. Hence, the influence of viscous boundary layers is less significant as turbine size increases, since the boundary layers occupy a less significant percentage of the flow path through the runner. In addition, the relative roughness ($\varepsilon/D$) on the surfaces of the prototype runner blades may be significantly smaller than that on the model turbine unless the model surfaces are micropolished. Finally, large full scale turbines have smaller tip clearances relative to blade diameter; therefore tip losses and leakage are less significant.

***Discussion*** The increase in efficiency between the model and prototype is significant, and helps us to understand why very large hydroturbines can be extremely efficient devices.

---

**14-108**

**Solution** We are to design a new hydroturbine by scaling up an existing hydroturbine. Specifically we are to calculate the new turbine diameter, volume flow rate, and brake horsepower.

***Assumptions*** **1** The new turbine is geometrically similar to the existing turbine. **2** Reynolds number effects and roughness effects are negligible. **3** The new penstock is also geometrically similar to the existing penstock so that the flow entering the new turbine (velocity profile, turbulence intensity, etc.) is similar to that of the existing turbine.

***Properties*** The density of water at 20°C is $\rho$ = 998.0 kg/m³.

***Analysis*** Since the new turbine (B) is dynamically similar to the existing turbine (A), we are concerned with only one particular homologous operating point of both turbines, namely the best efficiency point. We solve Eq. 14-38b for $D_B$,

$$D_B = D_A\sqrt{\frac{H_B}{H_A}}\left(\frac{\dot{n}_A}{\dot{n}_B}\right) = (1.50\text{ m})\sqrt{\frac{110\text{ m}}{90.0\text{ m}}}\left(\frac{150\text{ rpm}}{120\text{ rpm}}\right) = 2.0729 \cong \mathbf{2.07\ m}$$

We then solve Eq. 14-38a for $\dot{V}_B$,

$$\dot{V}_B = \dot{V}_A\left(\frac{\dot{n}_B}{\dot{n}_A}\right)\left(\frac{D_B}{D_A}\right)^3 = (162\text{ m}^3/\text{s})\left(\frac{120\text{ rpm}}{150\text{ rpm}}\right)\left(\frac{2.0729\text{ m}}{1.50\text{ m}}\right)^3 = 342.027\frac{\text{m}^3}{\text{s}} \cong \mathbf{342\ \frac{m^3}{s}}$$

Finally, we solve Eq. 14-38c for $bhp_B$,

$$bhp_B = bhp_A\left(\frac{\rho_B}{\rho_A}\right)\left(\frac{\dot{n}_B}{\dot{n}_A}\right)^3\left(\frac{D_B}{D_A}\right)^5 = 132\text{ MW}\left(\frac{998.0\text{ kg/m}^3}{998.0\text{ kg/m}^3}\right)\left(\frac{120\text{ rpm}}{150\text{ rpm}}\right)^3\left(\frac{2.0729\text{ m}}{1.50\text{ m}}\right)^5 = 340.62\text{ MW} \cong \mathbf{341\ MW}$$

***Discussion*** To avoid round-off errors, we save several more significant digits for $D_B$ than are given in the final answer.

## 14-109

**Solution** We are to compare the efficiency of two similar turbines, and discuss the Moody efficiency correction.

***Analysis*** We calculate the turbine efficiency for both turbines,

$$\eta_{\text{turbine A}} = \frac{bhp_A}{\rho_A g H_A \dot{V}_A} = \frac{132\times10^6\ \text{W}}{\left(998.0\ \text{kg/m}^3\right)\left(9.81\ \text{m/s}^2\right)\left(90\ \text{m}\right)\left(162\ \text{m}^3/\text{s}\right)}\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right) = \mathbf{92.5\%}$$

$$\eta_{\text{turbine B}} = \frac{bhp_B}{\rho_B g H_B \dot{V}_B} = \frac{340.62\times10^6\ \text{W}}{\left(998.0\ \text{kg/m}^3\right)\left(9.81\ \text{m/s}^2\right)\left(110\ \text{m}\right)\left(342.027\ \text{m}^3/\text{s}\right)}\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right) = \mathbf{92.5\%}$$

As expected, the two efficiencies are the same, since we have assumed dynamic similarity. However, total dynamic similarity may not actually be achieved between the two turbines because of scale effects (larger turbines generally have higher efficiency). The diameter of the new turbine is about 38% greater than that of the existing turbine, so the increase in efficiency due to turbine size should not be very significant. We verify this by using the Moody efficiency correction equation, considering turbine A as the "model" and B as the "prototype",

*Efficiency correction*:

$$\eta_{\text{turbine, B}} \approx 1-\left(1-\eta_{\text{turbine, A}}\right)\left(\frac{D_A}{D_B}\right)^{1/5} = 1-\left(1-0.925\right)\left(\frac{1.50\ \text{m}}{2.07\ \text{m}}\right)^{1/5} = \mathbf{0.930}$$

or **93.0%**. So, the first-order correction yields a predicted efficiency for the larger turbine that is about half of a percent greater than the smaller turbine. However, as mentioned in the text, we expect only about 2/3 of this increase, or 2(0.5%)/3 = 0.3%. Thus, our best estimate of the actual efficiency of the prototype is $\eta_{\text{turbine B}} \approx 92.5 + 0.3 = \mathbf{92.8\%}$.

Thus, our best estimate indicates that the new larger turbine will be slightly more efficient than its smaller brother, but the increase is only about 0.3%.

***Discussion*** If the flow entering the new turbine from the penstock were not similar to that of the existing turbine (e.g., velocity profile and turbulence intensity), we could not expect exact dynamic similarity.

---

## 14-110

**Solution** The turbine specific speed of two dynamically similar turbines is to be compared, and the most likely type of turbine is to be determined.

***Properties*** The density of water at $T$ = 20°C is $\rho$ = 998.0 kg/m$^3$.

***Analysis*** We calculate the dimensionless turbine specific speed for turbines A and B,

$$N_{St,A} = \frac{\omega_A \left(bhp_A\right)^{1/2}}{\left(\rho_A\right)^{1/2}\left(gH_A\right)^{5/4}} = \frac{\left(15.71\ \text{rad/s}\right)\left(132\times10^6\ \text{W}\right)^{1/2}}{\left(998.0\ \text{kg/m}^3\right)^{1/2}\left[\left(9.81\ \text{m/s}^2\right)\left(90.0\ \text{m}\right)\right]^{5/4}}\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right)^{1/2} = \mathbf{1.19}$$

$$N_{St,B} = \frac{\omega_B \left(bhp_B\right)^{1/2}}{\left(\rho_B\right)^{1/2}\left(gH_B\right)^{5/4}} = \frac{\left(12.57\ \text{rad/s}\right)\left(341\times10^6\ \text{W}\right)^{1/2}}{\left(998.0\ \text{kg/m}^3\right)^{1/2}\left[\left(9.81\ \text{m/s}^2\right)\left(110\ \text{m}\right)\right]^{5/4}}\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right)^{1/2} = \mathbf{1.19}$$

We see that the turbine specific speed of the two turbines is the same. In customary US units,

$$N_{St,\text{US,A}} = N_{St,\text{US,B}} = 43.46 N_{St} = \left(43.46\right)\left(1.19\right) = \mathbf{51.6}$$

From Fig. 14-108, both of these turbines are most likely **Francis turbines**.

***Discussion*** Since turbine A and turbine B operate at homologous points, it is no surprise that their turbine specific speeds are the same. In fact, if they weren't the same, it would be a sure sign of an algebraic or calculation error.

---

## Review Problems

### 14-111C
**Solution**

(*a*) *True*: As the gears turn, they direct a closed volume of fluid from the inlet to the outlet of the gear pump.

(*b*) *True* or *False*: Rotary pumps can be either positive displacement or dynamic (an unfortunate use of terminology). As a positive-displacement pump, the rotors direct a closed volume of fluid from the inlet to the outlet of the rotary pump. As a dynamic pump, "rotary pump" is sometimes used in place of the more correct term, "rotodynamic pump".

(*c*) *True*: At a given rotational speed, the volume flow rate of a positive-displacement pump is fairly constant regardless of load because of the fixed closed volume.

(*d*) *False*: Actually, the net head *increases* with fluid viscosity, because high viscosity fluids cannot penetrate the gaps as easily.

### 14-112C
**Solution** We are to discuss a water meter from the point of view of a piping system.

***Analysis*** Although a water meter is a type of turbine, when analyzing pipe flow systems, we would think of the water meter as **a type of minor loss in the system**, just as a valve, elbow, etc. would be a minor loss, since there is a pressure drop associated with flow through the water meter.

***Discussion*** In fact, manufacturers of water meters provide minor loss coefficients for their products.

### 14-113C
**Solution** We are to discuss the purpose of pump and turbine specific speeds.

***Analysis*** Pump specific speed is used to characterize the operation of a pump at its optimum conditions (best efficiency point), and is useful for **preliminary pump selection**. Likewise, turbine specific speed is used to characterize the operation of a turbine at its optimum conditions (best efficiency point), and is useful for **preliminary turbine selection.**

***Discussion*** Pump specific speed and turbine specific speed are parameters that can be calculated quickly. Based on the value obtained, one can quickly select the type of pump or turbine that is most appropriate for the given application.

### 14-114C
**Solution** We are to discuss the definition and usefulness of a pump-turbine.

***Analysis*** A pump-turbine is **a turbomachine that can run both as a *pump* and as a *turbine*** (by running in the opposite direction). A pump-turbine is used by some power plants for **energy storage**; specifically, water is pumped by the pump-turbine during periods of low demand for power, and electricity is generated by the pump-turbine during periods of high demand for power.

***Discussion*** We note that energy is "lost" both ways – when the pump-turbine is acting as a pump, and when it is acting as a turbine. Nevertheless, the energy storage scheme may still be cost-effective and profitable, in spite of the energy losses, because it may enable a power company to delay construction of costly new power-production facilities.

**14-115**

**Solution** We are to prove a relationship between two dynamically similar pumps, and discuss its application to turbines.

***Assumptions*** **1** The two pumps are geometrically similar. **2** Reynolds number and roughness effects are not critical in the analysis (the two pumps are not grossly different from each other in size, capacity, etc.).

***Analysis*** Since the two pumps are dynamically similar, dimensionless pump parameter $C_H$ must be the same for both pumps,

$$C_{H,\mathrm{A}} = \frac{gH_\mathrm{A}}{\omega_\mathrm{A}{}^2 D_\mathrm{A}{}^2} = C_{\mathrm{H,B}} = \frac{gH_\mathrm{B}}{\omega_\mathrm{B}{}^2 D_\mathrm{B}{}^2} \quad \rightarrow \quad \frac{\omega_\mathrm{A}}{\omega_\mathrm{B}} = \sqrt{\frac{H_\mathrm{A}}{H_\mathrm{B}}}\frac{D_\mathrm{B}}{D_\mathrm{A}}$$

Similarly, dimensionless pump parameter $C_Q$ must be the same for both pumps,

$$C_{Q,\mathrm{A}} = \frac{\dot{V}_\mathrm{A}}{\omega_\mathrm{A} D_\mathrm{A}{}^3} C_{\mathrm{Q,B}} = \frac{\dot{V}_\mathrm{B}}{\omega_\mathrm{B} D_\mathrm{B}{}^3} \quad \rightarrow \quad \left(\frac{D_\mathrm{B}}{D_\mathrm{A}}\right)^3 = \frac{\omega_\mathrm{A}}{\omega_\mathrm{B}}\frac{\dot{V}_\mathrm{B}}{\dot{V}_\mathrm{A}}$$

Combining the above two equations yields

$$\left(\frac{D_\mathrm{B}}{D_\mathrm{A}}\right)^3 = \sqrt{\frac{H_\mathrm{A}}{H_\mathrm{B}}}\frac{D_\mathrm{B}}{D_\mathrm{A}}\frac{\dot{V}_\mathrm{B}}{\dot{V}_\mathrm{A}}$$

which reduces to

$$\left(\frac{D_\mathrm{B}}{D_\mathrm{A}}\right)^2 = \sqrt{\frac{H_\mathrm{A}}{H_\mathrm{B}}}\frac{\dot{V}_\mathrm{B}}{\dot{V}_\mathrm{A}} \quad \rightarrow \quad \boxed{D_\mathrm{B} = D_\mathrm{A}\left(\frac{H_\mathrm{A}}{H_\mathrm{B}}\right)^{1/4}\left(\frac{\dot{V}_\mathrm{B}}{\dot{V}_\mathrm{A}}\right)^{1/2}}$$

Thus, we have eliminated the angular velocity as a parameter, and the relationship is proven.

Since turbines follow the same affinity laws as pumps, **the relationship also applies to two dynamically similar turbines**.

***Discussion*** In like manner, we can eliminate other parameters through algebraic manipulations to scale a pump or turbine up or down.

**14-116**

**Solution** We are to prove a relationship between two dynamically similar turbines, and discuss its application to pumps.

***Assumptions*** **1** The two turbines are geometrically similar. **2** Reynolds number and roughness effects are not critical in the analysis (the two turbines are not grossly different from each other in size, capacity, etc.).

***Analysis*** Since the two turbines are dynamically similar, dimensionless turbine parameter $C_H$ must be the same for both turbines,

$$C_{H,\text{A}} = \frac{gH_\text{A}}{\omega_\text{A}^{\ 2} D_\text{A}^{\ 2}} = C_{\text{H,B}} = \frac{gH_\text{B}}{\omega_\text{B}^{\ 2} D_\text{B}^{\ 2}} \quad \rightarrow \quad \frac{\omega_\text{A}}{\omega_\text{B}} = \sqrt{\frac{H_\text{A}}{H_\text{B}}} \frac{D_\text{B}}{D_\text{A}}$$

Similarly, dimensionless turbine parameter $C_P$ must be the same for both turbines,

$$C_{P,\text{A}} = \frac{bhp_\text{A}}{\rho_\text{A} \omega_\text{A}^{\ 3} D_\text{A}^{\ 5}} = C_{Q,\text{B}} = \frac{bhp_\text{B}}{\rho_\text{B} \omega_\text{B}^{\ 3} D_\text{B}^{\ 5}} \quad \rightarrow \quad \left(\frac{D_\text{B}}{D_\text{A}}\right)^5 = \left(\frac{\omega_\text{A}}{\omega_\text{B}}\right)^3 \frac{\rho_\text{A}}{\rho_\text{B}} \frac{bhp_\text{B}}{bhp_\text{A}}$$

Combining the above two equations yields

$$\left(\frac{D_\text{B}}{D_\text{A}}\right)^5 = \left(\frac{H_\text{A}}{H_\text{B}}\right)^{3/2} \left(\frac{D_\text{B}}{D_\text{A}}\right)^3 \frac{\rho_\text{A}}{\rho_\text{B}} \frac{bhp_\text{B}}{bhp_\text{A}}$$

which reduces to

$$\left(\frac{D_\text{B}}{D_\text{A}}\right)^2 = \left(\frac{H_\text{A}}{H_\text{B}}\right)^{3/2} \frac{\rho_\text{A}}{\rho_\text{B}} \frac{bhp_\text{B}}{bhp_\text{A}} \quad \rightarrow \quad \boxed{D_\text{B} = D_\text{A} \left(\frac{H_\text{A}}{H_\text{B}}\right)^{3/4} \left(\frac{\rho_\text{A}}{\rho_\text{B}}\right)^{1/2} \left(\frac{bhp_\text{B}}{bhp_\text{A}}\right)^{1/2}}$$

Thus, we have eliminated the angular velocity as a parameter, and the relationship is proven.

Since pumps follow the same affinity laws as turbines, **the relationship also applies to two dynamically similar pumps**.

***Discussion*** In like manner, we can eliminate other parameters through algebraic manipulations to scale a pump or turbine up or down.

## Design and Essay Problems

**14-117** [*Also solved using EES on enclosed DVD*]

**Solution** We are to generate a computer application that uses the affinity laws to design a new pump that is dynamically similar to an existing pump.

***Assumptions*** **1** The two pumps are geometrically similar. **2** Reynolds number and roughness effects are not critical in the analysis (the two pumps are not grossly different from each other in size, capacity, etc.).

***Analysis*** First, we calculate the brake horsepower for pump A,

$$bhp_A = \frac{\rho_A \dot{V}_A g H_A}{\eta_{\text{pump,A}}} = \frac{\left(998.0\text{ kg/m}^3\right)\left(0.00040\text{ m}^3\text{/s}\right)\left(9.81\text{ m/s}^2\right)\left(1.20\text{ m}\right)}{0.81}\left(\frac{\text{W}\cdot\text{s}^3}{\text{kg}\cdot\text{m}^2}\right) = \mathbf{5.80\ W}$$

We use the affinity laws to calculate the new pump parameters. Using the given data, we first calculate the new pump diameter,

$$D_B = D_A\left(\frac{H_A}{H_B}\right)^{1/4}\left(\frac{\dot{V}_B}{\dot{V}_A}\right)^{1/2} = \left(5.0\text{ cm}\right)\left(\frac{120\text{ cm}}{450\text{ cm}}\right)^{1/4}\left(\frac{2400\text{ cm}^3\text{/s}}{400\text{ cm}^3\text{/s}}\right)^{1/2} = \mathbf{8.80\ cm}$$

Knowing $D_B$, we calculate the rotation rate of the new pump,

$$\dot{n}_B = \dot{n}_A\frac{D_A}{D_B}\left(\frac{H_B}{H_A}\right)^{1/2} = \left(1725\text{ rpm}\right)\frac{5.0\text{ cm}}{8.80\text{ cm}}\left(\frac{450\text{ cm}}{120\text{ cm}}\right)^{1/2} = \mathbf{1898\ rpm}$$

Knowing $D_B$ and $\omega_B$, we now calculate the required shaft power,

$$bhp_B = bhp_A\frac{\rho_B}{\rho_A}\left(\frac{D_B}{D_A}\right)^5\left(\frac{\omega_B}{\omega_A}\right)^3 = \left(5.80\text{ W}\right)\frac{1226\text{ kg/m}^3}{998.0\text{ kg/m}^3}\left(\frac{8.80\text{ cm}}{5.0\text{ cm}}\right)^5\left(\frac{1898\text{ rpm}}{1725\text{ rpm}}\right)^3 = \mathbf{160\ W}$$

Our answers agree with those given. Now we are confident that we can apply our computer program to other design problems of similar nature, as in the next problem.

***Discussion*** To avoid round-off errors in the calculations, we saved several more significant digits for $D_B$ than are reported here.



**14-118** 

**Solution** We are to use a computer code and the affinity laws to design a new pump that is dynamically similar to an existing pump.

***Assumptions*** **1** The two pumps are geometrically similar. **2** Reynolds number and roughness effects are not critical in the analysis (the two pumps are not grossly different from each other in size, capacity, etc.). **3** Both pumps operate at their BEP.

***Analysis*** Using our computer code, $D_B$ = **12.8 cm**, $\dot{n}_B$ = **1924 rpm**, and $bhp_B$ = **235 W**. We calculate pump specific speed for the new pump,

*Pump B*:
$$N_{Sp} = \frac{\omega \dot{V}^{1/2}}{(gH)^{3/4}} = \frac{(1924 \text{ rpm})\left(\frac{2\pi}{60}\frac{\text{rad/s}}{\text{rpm}}\right)\left(0.00367 \text{ m}^3\text{/s}\right)^{1/2}}{\left((9.81 \text{ m/s}^2)(5.70 \text{ m})\right)^{3/4}} = \mathbf{0.597}$$

and we repeat the calculation for the existing pump,

*Pump A*:
$$N_{Sp} = \frac{\omega \dot{V}^{1/2}}{(gH)^{3/4}} = \frac{(1500 \text{ rpm})\left(\frac{2\pi}{60}\frac{\text{rad/s}}{\text{rpm}}\right)\left(0.00135 \text{ m}^3\text{/s}\right)^{1/2}}{\left((9.81 \text{ m/s}^2)(2.10 \text{ m})\right)^{3/4}} = \mathbf{0.597}$$

Our answers agree, as they must, since the two pumps are operating at homologous points. From Fig. 14-73, we can see that these are most likely **centrifugal pumps**.

***Discussion*** The pump performance parameters of pump B can be calculated manually to further verify our computer code.



**14-119** 

**Solution** We are to generate a computer application that uses the affinity laws to design a new turbine that is dynamically similar to an existing turbine.

***Assumptions*** **1** The two turbines are geometrically similar. **2** Reynolds number and roughness effects are not critical in the analysis (the two turbines are not grossly different from each other in size, capacity, etc.).

***Analysis*** We use the affinity laws to calculate the new turbine parameters. Using the given data, we first calculate the new turbine diameter,

$$D_B = D_A\sqrt{\frac{H_B}{H_A}}\left(\frac{\dot{n}_A}{\dot{n}_B}\right) = (1.40\text{ m})\sqrt{\frac{95.0\text{ m}}{80.0\text{ m}}}\left(\frac{150\text{ rpm}}{120\text{ rpm}}\right) = \mathbf{1.91\text{ m}}$$

We then solve Eq. 14-38a for $\dot{V}_B$,

$$\dot{V}_B = \dot{V}_A\left(\frac{\dot{n}_B}{\dot{n}_A}\right)\left(\frac{D_B}{D_A}\right)^3 = \left(162\text{ m}^3/\text{s}\right)\left(\frac{120\text{ rpm}}{150\text{ rpm}}\right)\left(\frac{1.91\text{ m}}{1.40\text{ m}}\right)^3 = \mathbf{328\text{ m}^3/\text{s}}$$

Finally, we solve Eq. 14-38c for $bhp_B$,

$$bhp_B = bhp_A\left(\frac{\rho_B}{\rho_A}\right)\left(\frac{\dot{n}_B}{\dot{n}_A}\right)^3\left(\frac{D_B}{D_A}\right)^5 = 118\text{ MW}\left(\frac{998.0\text{ kg/m}^3}{998.0\text{ kg/m}^3}\right)\left(\frac{120\text{ rpm}}{150\text{ rpm}}\right)^3\left(\frac{1.91\text{ m}}{1.40\text{ m}}\right)^5 = \mathbf{283\text{ MW}}$$

Our answers agree with those given. Now we are confident that we can apply our computer program to other design problems of similar nature.

***Discussion*** To avoid round-off errors in the calculations, we saved several more significant digits for $D_B$ than are reported here.

---

**14-120** 

**Solution** We are to use a computer program and the affinity laws to design a new turbine that is dynamically similar to an existing turbine.

***Assumptions*** **1** The two pumps are geometrically similar. **2** Reynolds number and roughness effects are not critical in the analysis (the two pumps are not grossly different from each other in size, capacity, etc.).

***Analysis*** Using our computer code, $D_B$ = **2.04 m**, $\dot{V}_B$ = 815 m³/s, and $bhp_B$ = **577 MW**. We calculate turbine specific speed for the new turbine,

$$N_{St,A} = \frac{\omega_A\left(bhp_A\right)^{1/2}}{\left(\rho_A\right)^{1/2}\left(gH_A\right)^{5/4}} = \frac{\left(25.13\text{ rad/s}\right)\left(11.4\times10^6\text{ W}\right)^{1/2}}{\left(998.0\text{ kg/m}^3\right)^{1/2}\left[\left(9.81\text{ m/s}^2\right)\left(22.0\text{ m}\right)\right]^{5/4}}\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right)^{1/2} = \mathbf{3.25}$$

and for turbine B,

$$N_{St,B} = \frac{\omega_B\left(bhp_B\right)^{1/2}}{\left(\rho_B\right)^{1/2}\left(gH_B\right)^{5/4}} = \frac{\left(21.99\text{ rad/s}\right)\left(577\times10^6\text{ W}\right)^{1/2}}{\left(998.0\text{ kg/m}^3\right)^{1/2}\left[\left(9.81\text{ m/s}^2\right)\left(95.0\text{ m}\right)\right]^{5/4}}\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right)^{1/2} = \mathbf{3.25}$$

Our answers agree, as they must, since the two turbines are operating at homologous points. From Fig. 14-108, we see that these are most likely **Kaplan turbines**.

***Discussion*** The turbine parameters of turbine B can be calculated manually to further verify our computer code.

---

**14-121**

**Solution** We are to compare efficiencies for two geometrically similar turbines, and discuss the Moody efficiency correction.

***Analysis*** We calculate the turbine efficiency for both turbines,

$$\eta_{\text{turbine A}} = \frac{bhp_{\text{A}}}{\rho_{\text{A}} g H_{\text{A}} \dot{V}_{\text{A}}} = \frac{11.4\times10^6 \text{ W}}{\left(998.0 \text{ kg/m}^3\right)\left(9.81 \text{ m/s}^2\right)\left(22.0 \text{ m}\right)\left(69.5 \text{ m}^3/\text{s}\right)}\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right) = \mathbf{76.2\%}$$

and

$$\eta_{\text{turbine B}} = \frac{bhp_{\text{B}}}{\rho_{\text{B}} g H_{\text{B}} \dot{V}_{\text{B}}} = \frac{577\times10^6 \text{ W}}{\left(998.0 \text{ kg/m}^3\right)\left(9.81 \text{ m/s}^2\right)\left(95.0 \text{ m}\right)\left(814.6 \text{ m}^3/\text{s}\right)}\left(\frac{\text{kg}\cdot\text{m}^2}{\text{W}\cdot\text{s}^3}\right) = \mathbf{76.2\%}$$

where we have included an extra digit in the intermediate values to avoid round-off error. As expected, the two efficiencies are the same, since we have assumed dynamic similarity. However, total dynamic similarity may not actually be achieved between the two turbines because of scale effects (larger turbines generally have higher efficiency). The diameter of the new turbine is more than twice that of the existing turbine, so the increase in efficiency due to turbine size may be significant. We account for the size increase by using the Moody efficiency correction equation, considering turbine A as the "model" and B as the "prototype",

*Efficiency correction*:
$$\eta_{\text{turbine, B}} \approx 1-\left(1-\eta_{\text{turbine, A}}\right)\left(\frac{D_{\text{A}}}{D_{\text{B}}}\right)^{1/5} = 1-\left(1-0.762\right)\left(\frac{0.86 \text{ m}}{2.04 \text{ m}}\right)^{1/5} = \mathbf{0.800}$$

or **80.0%**. So, the first-order correction yields a predicted efficiency for the larger turbine that is about four percent greater than the smaller turbine. However, as mentioned in the text, we expect only about 2/3 of this increase, or 2(0.800 – 0.762)/3 = 0.025 or 2.5%. Thus, our best estimate of the actual efficiency of the prototype is

$$\eta_{\text{turbine B}} \approx 76.2 + 2.5 = \mathbf{78.7\%}$$

The higher efficiency of the new larger turbine is significant because an increase in power production of 2.5% can lead to significant profits for the power company.

***Discussion*** If the flow entering the new turbine from the penstock were not similar to that of the existing turbine (e.g., velocity profile and turbulence intensity), we could not expect exact dynamic similarity.



# Solutions Manual for

# Fluid Mechanics: Fundamentals and Applications

# by Çengel & Cimbala

---

# CHAPTER 15

# INTRODUCTION TO COMPUTATIONAL FLUID DYNAMICS

---

## Fundamentals, Grid Generation, and Boundary Conditions

### 15-1C

**Solution** We are to list the unknowns and the equations for a given flow situation.

***Analysis*** There are only three unknowns in this problem, ***u*, *v*, and *P* (or*P'*)**. Thus, we require three equations: **continuity**, ***x* momentum** (or *x* component of Navier-Stokes), and ***y* momentum** (or *y* component of Navier-Stokes). These equations, when combined with the appropriate boundary conditions, are sufficient to solve the problem.

***Discussion*** The actual equations to be solved by the computer are discretized versions of the differential equations.

### 15-2C

**Solution** We are to define several terms or phrases and provide examples.

***Analysis***

(*a*) **A computational domain is a region in space (either 2-D or 3-D) in which the numerical equations of fluid flow are solved by CFD**. The computational domain is bounded by edges (2-D) or faces (3-D) on which boundary conditions are applied.

(*b*) **A mesh is generated by dividing the computational domain into tiny cells**. The numerical equations are then solved in each cell of the mesh. A mesh is also called a grid.

(*c*) **A transport equation is a differential equation representing how some property is transported through a flow field**. The transport equations of fluid mechanics are conservation equations. For example, the continuity equation is a differential equation representing the transport of mass, and also conservation of mass. The Navier-Stokes equation is a differential equation representing the transport of linear momentum, and also conservation of linear momentum.

(*d*) **Equations are said to be coupled when at least one of the variables (unknowns) appears in more than one equation**. In other words, the equations cannot be solved alone, but must be solved simultaneously with each other. This is the case with fluid mechanics since each component of velocity, for example, appears in the continuity equation and in all three components of the Navier-Stokes equation.

***Discussion*** Students' definitions should be in their own words.

### 15-3C

**Solution** We are to discuss the difference between nodes and intervals and analyze a given computational domain in terms of nodes and intervals.

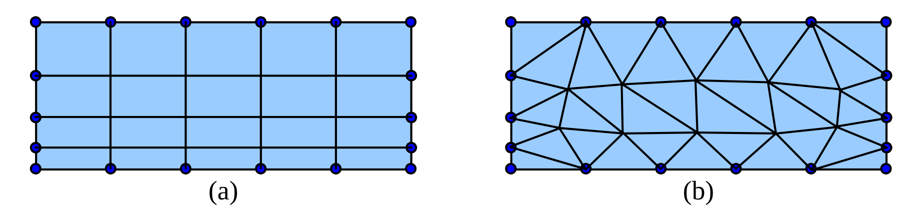
***Analysis*** **Nodes are points along an edge of a computational domain that represent the vertices of cells**. In other words, they are the points where corners of the cells meet. **Intervals, on the other hand, are short line segments between nodes**. Intervals represent the small edges of cells themselves. In Fig. P15-3 there are **6 nodes** and **5 intervals** on the top and bottom edges. There are **5 nodes** and **4 intervals** on the left and right edges.

***Discussion*** We can extend the node and interval concept to three dimensions.

## 15-4C

**Solution** For a given computational domain with specified nodes and intervals we are to compare a structured grid and an unstructured grid and discuss.

***Analysis*** We construct the two grids in the figure: (a) structured, and (b) unstructured.




There are 5 × 4 = 20 cells in the structured grid, and there are 36 cells in the unstructured grid.

***Discussion*** Depending on how individual students construct their unstructured grid, the shape, size, and number of cells may differ considerably.

---

## 15-5C

**Solution** We are to summarize the eight steps involved in a typical CFD analysis.

***Analysis*** We list the steps in the order presented in this chapter:

1. **Specify a computational domain and generate a grid.**
2. **Specify boundary conditions on all edges or faces.**
3. **Specify the type of fluid and its properties.**
4. **Specify numerical parameters and solution algorithms.**
5. **Apply initial conditions as a starting point for the iteration.**
6. **Iterate towards a solution.**
7. **After convergence, analyze the results (post processing).**
8. **Calculate global and integral properties as needed.**

***Discussion*** The order of some of the steps is interchangeable, particularly Steps 2 through 5.

---

## 15-6C

**Solution** We are to explain why the cylinder should not be centered horizontally in the computational domain.

***Analysis*** Flow separates over bluff bodies, generating a wake with **reverse flow and eddies downstream of the body**. There are no such problems upstream. Hence it is always wise to extend the downstream portion of the domain as far as necessary to avoid reverse flow problems at the outlet boundary.

***Discussion*** The same problems arise at the outlet of ducts and pipes – sometimes we need to extend the duct to avoid reverse flow at the outlet boundary.

---

**15-7C**

**Solution** We are to discuss the significance of several items with respect to iteration.

***Analysis***

(*a*) In a CFD solution, we typically iterate towards a solution. **In order to get started, we make some initial conditions for all the variables (unknowns) in the problem**. These initial conditions are wrong, of course, but they are necessary as a starting point. Then we begin the iteration process, eventually obtaining the solution.

(*b*) **A residual is a measure of how much our variables differ from the "exact" solution**. We construct a residual by putting all the terms of a transport equation on one side, so that the terms all add to zero if the solution is correct. As we iterate, the terms will *not* add up to zero, and the remainder is called the residual. As the CFD solution iterates further, the residual should (hopefully) decrease.

(*c*) **Iteration is the numerical process of marching towards a final solution**, beginning with initial conditions, and progressively correcting the solution. As the iteration proceeds, the variables converge to their final solution as the residuals decrease.

(*d*) **Once the CFD solution has converged, post processing is performed on the solution**. Examples include plotting velocity and pressure fields, calculating global properties, generating other flow quantities like vorticity, etc. Post processing is performed after the CFD solution has been found, and does not change the results. Post processing is generally not as CPU intensive as the iterative process itself.

***Discussion*** We have assumed steady flow in the above discussions.

---

**15-8C**

**Solution** We are to discuss how the iteration process is made faster.

***Analysis***

(*a*) With *multigridding*, **solutions of the equations of motion are obtained on a coarse grid first, followed by successively finer grids**. This speeds up convergence because the gross features of the flow are quickly established on the coarse grid, and then the iteration process on the finer grid requires less time.

(*b*) In some CFD codes, **a steady flow is treated as though it were an unsteady flow. Then, an *artificial time* is used to march the solution in time**. Since the solution is steady, however, the solution approaches the steady-state solution as "time" marches on. In some cases, this technique yields faster convergence.

***Discussion*** There are other "tricks" to speed up the iteration process, but CFD solutions often take a long time to converge.

---

**15-9C**

**Solution** We are to list the boundary conditions that are applicable to a given edge, and we are to explain why other boundary conditions are not applicable.

***Analysis*** We may apply the following boundary conditions: **outflow**, **pressure inlet**, **pressure outlet**, **symmetry** (to be discussed), **velocity inlet**, and **wall**. The curved edge cannot be an *axis* because an axis must be a straight line. The edge cannot be a *fan* or *interior* because such edges cannot be at the outer boundary of a computational domain. Finally, **the edge cannot be periodic since there is no other edge along the boundary of the computational domain that is of identical shape** (a periodic boundary must have a "partner"). The symmetry boundary condition merits further discussion. Numerically, gradients of flow variables in the direction normal to a symmetry boundary condition are set to zero, and there is no mathematical reason why the curved right edge of the present computational domain cannot be set as symmetry. However, you would be hard pressed to think of a physical situation in which a curved edge like that of Fig. P15-9 would be a valid symmetry boundary condition.

***Discussion*** Just because you can set a boundary condition and generate a CFD result does not guarantee that the result is physically meaningful.

---

**15-10C**

**Solution** We are to discuss the standard method to test for adequate grid resolution.

***Analysis*** The standard method to test for adequate grid resolution is to **increase the resolution** (by a factor of 2 in all directions if feasible) **and repeat the simulation**. If the results do not change appreciably, the original grid is deemed adequate. If, on the other hand, there are significant differences between the two solutions, the original grid is likely of inadequate resolution. In such a case, an even finer grid should be tried until the grid is adequately resolved.

***Discussion*** Keep in mind that if the boundary conditions are not specified properly, or if the chosen turbulence model is not appropriate for the flow being simulated by CFD, no amount of grid refinement is going to make the solution more physically correct.

---

**15-11C**

**Solution** We are to discuss the difference between a pressure inlet boundary condition and a velocity inlet boundary condition, and we are to explain why both pressure and velocity cannot be specified on the same boundary.

***Analysis*** **At a pressure inlet we specify the pressure but not the velocity**. **At a velocity inlet we specify the opposite – velocity but not pressure**. To specify both pressure and velocity would lead to mathematical **over-specification, since pressure and velocity are coupled in the equations of motion**. When pressure is specified at a pressure inlet (or outlet), the CFD code automatically adjusts the velocity at that boundary. In a similar manner, when velocity is specified at a velocity inlet, the CFD code adjusts the pressure at that boundary.

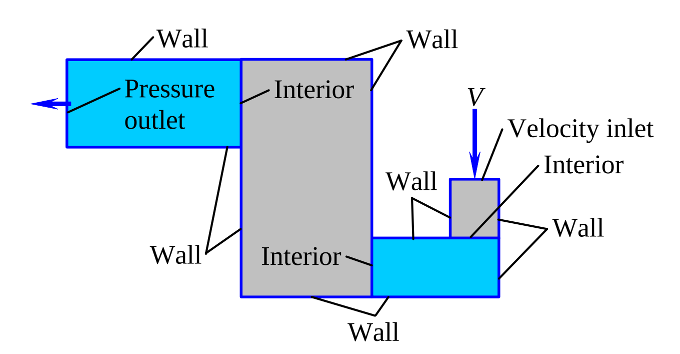
***Discussion*** Since pressure and velocity are coupled, specification of both at a boundary would lead to inconsistencies in the equations of motion at that boundary.

---

**15-12C**

**Solution** We are to label all the boundary conditions to be applied to a computational domain.

***Analysis*** The inlet is a *velocity inlet*. The outlet is a *pressure outlet*. All other edges that define the outer limits of the computational domain are *walls*. Finally, there are three edges that must be specified as *interior*. These are all labeled in the figure below.




***Discussion*** It is critical that each boundary condition be specified carefully. Otherwise the CFD solution will not be correct.

---

**15-13C**

**Solution** We are to analyze what will happen to inlet pressure and outlet velocity when a fan is turned on in the computational domain of the previous problem.

***Analysis*** Since the fan helps to push air through the channel, the inlet pressure will adjust itself so that less inlet pressure is required. In other words, the **inlet pressure will decrease when the fan is turned on**. Since the inlet velocity is the same in both cases, the mass flow rate (and volume flow rate since the flow is incompressible) must remain the same for either case. Therefore, **outlet velocity will not change**.

***Discussion*** It may seem at first glance that $V_{out}$ should increase because of the fan, but in order to conserve mass, the outlet velocity cannot change. The solution is constrained by the specified inlet boundary condition. In a real physical experiment, there is no such restriction. The fan would cause the inlet pressure to decrease, the inlet velocity to increase, and the outlet velocity to increase.

---

**15-14C**

**Solution** We are to list and briefly describe six boundary conditions, and we are to give an example of each.

***Analysis*** In the chapter we list ten, so any six of these will suffice:

- **Axis**: Used in axisymmetric flows as the axis of rotation. Example: the axis of a torpedo-shaped body.
- **Fan**: An internal edge (2-D) or face (3-D) across which a sudden pressure rise is specified. Example: an axial flow fan in a duct.
- **Interior**: An internal edge (2-D) or face (3-D) across which nothing special happens – the interior boundary condition is used at the interface between two blocks. Example: all of the multiblock problems in this chapter, which require this boundary condition at the interface between any two blocks.
- **Outflow**: An outlet boundary condition in which the gradient of fluid properties is zero normal to the outflow boundary. Outflow is typically useful far away from the object or area of interest in a flow field. Example: the far field of flow over a body.
- **Periodic**: When the physical geometry has periodicity, the periodic boundary condition is used to specify that whatever passes through one face of the periodic pair must simultaneously enter the other face of the periodic pair. Example: in a heat exchanger where there are several rows of tubes.
- **Pressure inlet**: An inflow boundary in which pressure (but not velocity) is known and specified across the face. Example: the high pressure settling chamber of a blow-down wind tunnel facility.
- **Pressure outlet**: An outflow boundary in which pressure (but not velocity) is known and specified across the face. Example: the outlet of a pipe exposed to atmospheric pressure.
- **Symmetry**: A face over which the gradients of all flow variables are set to zero normal to the face – the result is a mirror image across the symmetry plane. Fluid cannot flow *through* a symmetry plane. Example: the mid-plane of flow over a circular cylinder in which the lower half is a mirror image of the upper half.
- **Velocity inlet**: An inflow boundary condition in which velocity (but not pressure) is known and specified across the face. Example: a uniform freestream inlet flow entering a computational domain from one side.
- **Wall**: A boundary through which fluid cannot pass and at which the no-slip condition (or a shear stress condition) is applied. Example: the surface of an airfoil that is being modeled by CFD.

***Discussion*** There are additional boundary conditions used in CFD calculations, but these are the only ones discussed in this chapter.

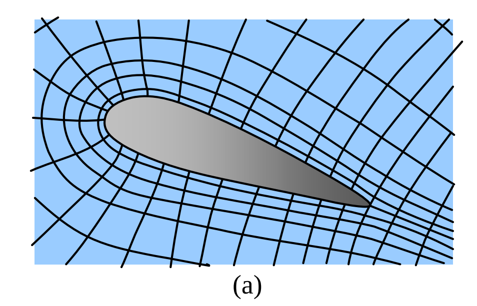
---

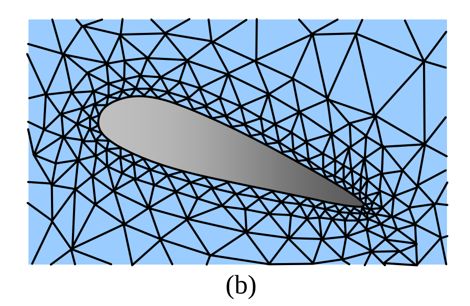
## 15-15

**Solution** We are to sketch a structured and an unstructured grid near the airfoil surface, and discuss advantages and disadvantages of each.

***Analysis*** In either case it is wise to cluster cells close to the airfoil surface since we expect that a thin boundary layer will exist along the surface, and we need many tiny cells within the boundary layer to adequately resolve it. Some simple, coarse meshes are drawn in Fig. 1. We would certainly want much higher resolution for CFD calculations.

**FIGURE 1**
A coarse structured (a) and unstructured (b) grid. Notice that the cells are clustered (more fine) near the surface of the airfoil since there is likely to be large velocity gradients there (in the boundary layer).


(a)


(b)

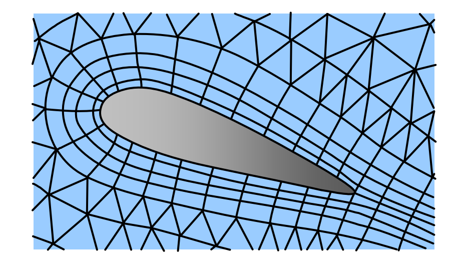
The structured grid in Fig 1a is called a C-grid since it wraps around the airfoil like the letter "C". The main advantage of the structured grid is that we can get high resolution near the surface with few cells. The main advantage of the unstructured grid is that it is somewhat easier to generate when the geometry is complicated (especially for highly curved surfaces). Furthermore, it is easier to transition between curved and straight edges with an unstructured grid. The main disadvantage of an unstructured grid is that more cells are required for the same spatial resolution.

***Discussion*** There are numerous other ways to construct a grid around this airfoil.

## 15-16

**Solution** We are to sketch a hybrid grid around an airfoil and explain its advantages.

***Analysis*** We sketch a hybrid grid in the figure. Note that the grid is structured near the airfoil surface, but unstructured beyond the surface. **The advantage of a hybrid grid is that it combines the advantages of both structured and unstructured grids**. Near surfaces we can use a structured grid to finely resolve the boundary layer with a minimum number of cells, and away from surfaces we can use an unstructured grid so that we can rapidly expand the cell size. We can also more easily blend the grid into the edges of the computational domain with an unstructured grid.



***Discussion*** A structured grid is generally the best choice, but a hybrid grid is often a better option than a fully unstructured grid.

## 15-17

**Solution** We are to sketch the blocking for a structured grid, sketch a coarse grid, and label all the boundary conditions to be applied to the computational domain.

***Analysis*** First of all, we recognize that because of symmetry, we can split the domain in half vertically. We construct four blocks around the half-cylinder to transform from round to square, and then we add simple rectangular blocks upstream and downstream of the cylinder (Fig. 1). There is a total of **six blocks**.

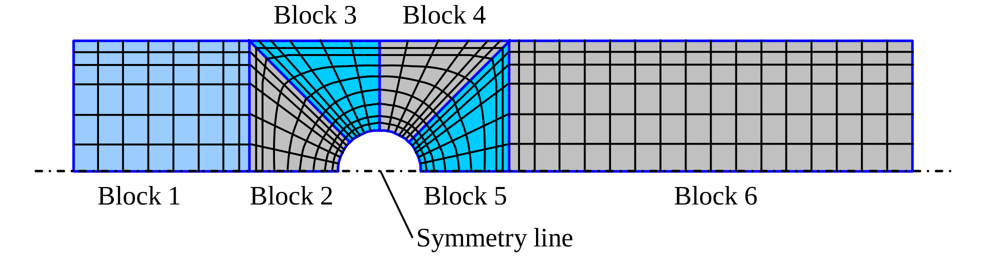



**FIGURE 1**
A possible blocking topology and coarse structured grid for a 2-D multiblock computational domain.

With the block structure of Fig. 1 no cells are highly skewed, and cells are clustered near the cylinder wall and the upper wall of the duct as desired.

The bottom edge of the computational domain is a line of **symmetry**. The inlet is a **velocity inlet**. The outlet is a **pressure outlet**. The upper edge of the computational domain is a **wall**. The edges that define the cylinder are also **walls**. Finally, there are 5 edges that are specified as **interior**. These are all labeled in Fig. 2.

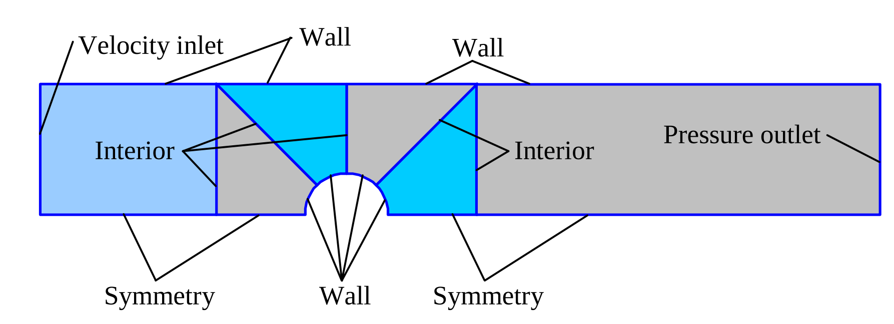



**FIGURE 2**
Boundary conditions specified on each edge of a 2-D multiblock computational domain.

***Discussion*** There are alternative ways to set up the blocking topology. For example, at the top we may define a thin block (Block 7) that stretches across the entire horizontal domain so that the boundary layer on the top wall of the channel can be more adequately resolved (Fig. 3).

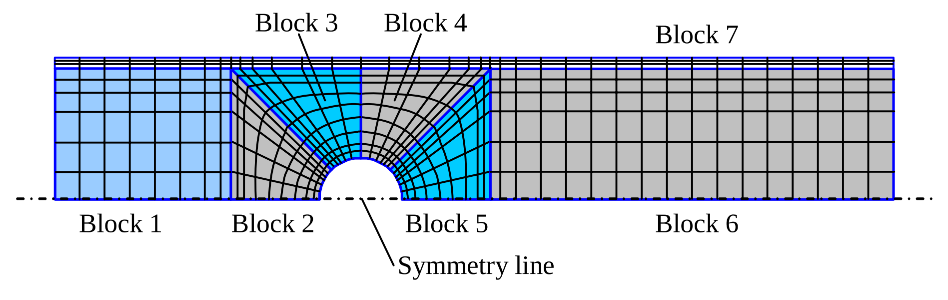



**FIGURE 3**
An alternative blocking topology and coarse structured grid for the 2-D multiblock computational domain. A seventh block is added at the top for better boundary layer resolution near the top plate.

**15-18**

**Solution** We are to sketch the blocking for a structured grid, sketch a coarse grid, and label all the boundary conditions to be applied to the computational domain.

***Analysis*** First of all, we recognize that because of symmetry, we can split the domain in half vertically. We construct four blocks inside the half-circle, and then we add blocks with one curved edge and three straight edges upstream and downstream of the cylinder (Fig. 1).

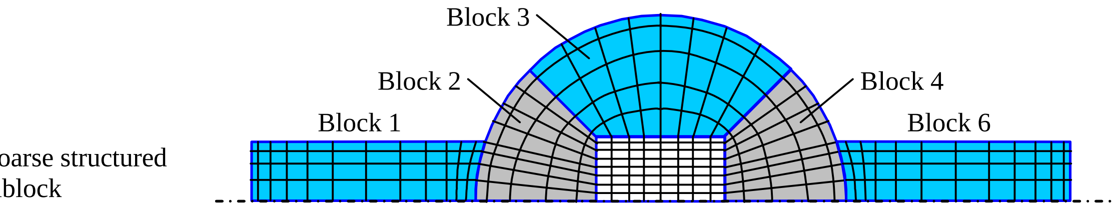



**FIGURE 1**
The blocking and coarse structured grid for a 2-D multiblock computational domain.

The setup of Fig. 1 contains six blocks. With this block structure, no cells are highly skewed, and cells are clustered near upper wall of the duct as desired. Cells are also clustered at the junctions between Blocks 1 and 2 and Blocks 4 and 6, where flow separation may occur.

The bottom edge of the computational domain is a line of **symmetry**. The inlet is a **velocity inlet**. The outlet is a **pressure outlet**. The upper edge of the computational domain is a **wall**. The edges that define the cylinder are also **walls**. Finally, there are 5 edges that are specified as **interior**. These are all labeled in Fig. 2.

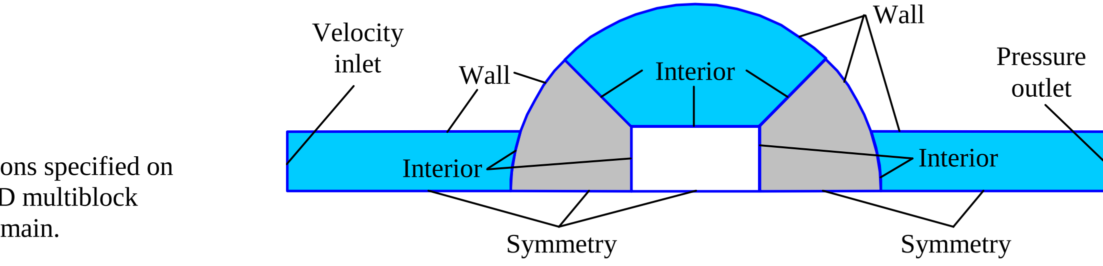



**FIGURE 2**
Boundary conditions specified on each edge of a 2-D multiblock computational domain.

***Discussion*** There are of course, alternative ways to set up the blocking topology.

## 15-19

**Solution** We are to modify an existing grid so that all blocks are elementary blocks. Then we are to verify that the total number of cells does not change.

***Analysis*** The right edge of Block 2 of Fig. 15-11b is split twice to accommodate Block 1. We therefore split Block 2 into three separate elementary blocks. Unfortunately, this process ends up splitting Block 6 twice, which in turn splits Block 4 twice. **We end up with 12 elementary blocks** as shown in Fig. 1.

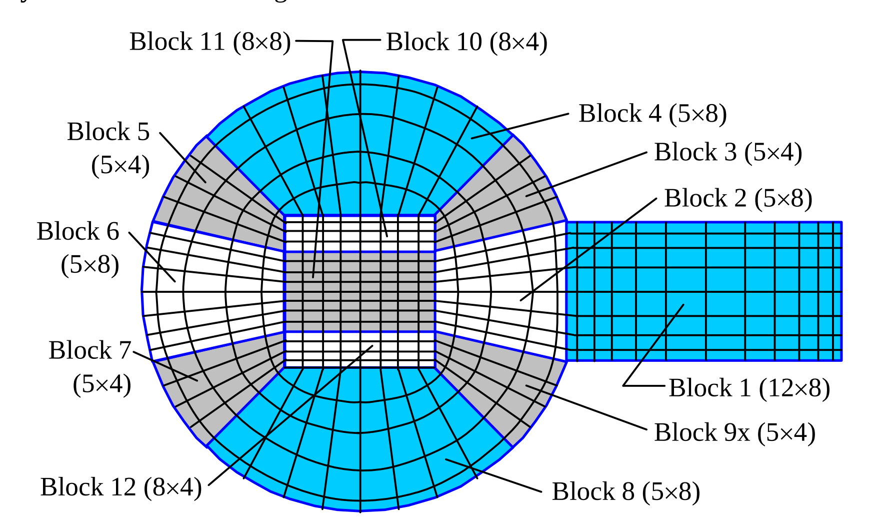



**FIGURE 1**
The blocking and coarse structured grid for a 2-D multiblock computational domain. Only elementary blocks are used in this grid.

We add up all the cells in these 12 blocks – we get a total of **464 cells**. This agrees with the total of 464 cells for the original 6 blocks in the domain.

***Discussion*** Sometimes it is easier to create a grid with elementary blocks, even if the CFD code can accept blocks with split edges or faces.

## 15-20

**Solution** We are to modify an existing grid into a smaller number of non- elementary blocks, and we are to verify that the total number of cells does not change.

***Analysis*** We combine Blocks 2, 3, 4, and 5 of Fig. 15-10b. Together, these produce one structured grid that wraps around the square in the middle – there are still 5 *i* intervals, but now there are 48 *j* intervals. **We end up with 3 non-elementary blocks**, as shown in Fig. 1.

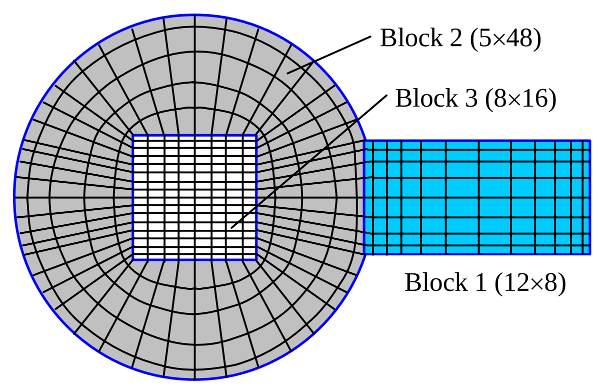



**FIGURE 1**
The blocking and coarse structured grid for a 2-D multiblock computational domain. Only elementary blocks are used in this grid.

We add up all the cells in these 12 blocks – we get a total of **464 cells**. This agrees with the total of 464 cells for the original 6 blocks in the domain.

***Discussion*** Block 2 in Fig. 1 is called an **O-grid** (for obvious reasons).

**15-21**

**Solution** We are to generate a computational domain and label all appropriate boundary conditions for one stage of a new heat exchanger design.

***Analysis*** We take advantage of the periodicity of the geometry. There are several ways to create a periodic grid for this flow. The simplest computational domain consists of a single flow passage between two neighboring tubes. We can make the periodic edge intercept anywhere on the front portion of the tube that we desire. We choose the lower surface for convenience and simplicity. The periodic computational domain is sketched in Fig. 1.

Boundary conditions are also straightforward, and are labeled in Fig. 1. For a known inlet velocity we set the boundary condition at the left edge as a **velocity inlet**. The tube walls are obviously set as **walls**. The outlet can be set as either a **pressure outlet** or an **outflow**, depending on the provided information and how far the outlet region extends beyond the tubes. Finally, we set two pairs of **translationally periodic** boundaries, one fore and one aft of the tubes. We label them separately to avoid confusion.

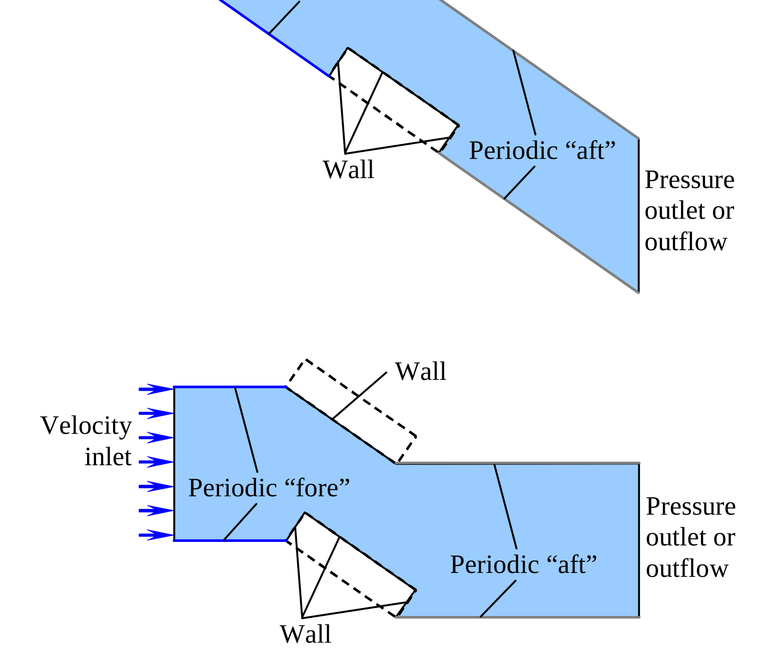



**FIGURE 1**
A periodic computational domain for a given geometry. Boundary conditions are also labeled.

**FIGURE 2**
An alternative periodic computational domain for a given geometry. Boundary conditions are also labeled.

***Discussion*** The fore and aft periodic edges are not horizontal in Fig. 1. This is not a problem since the periodic boundary condition is not restricted to horizontal or even to flat surfaces. An alternative, equally acceptable computational domain is shown in Fig. 2.

**15-22**

**Solution** We are to sketch a structured multiblock grid with four-sided elementary blocks for a given computational domain.

***Analysis*** We choose the computational domain of Fig. 1 of the previous problem. Since all edges are straight, the blocking scheme can be rather simple. We sketch the blocking topology and apply a coarse mesh in Fig. 1 for the case in which the CFD code does not require the node distribution to be exactly the same on periodic pairs.

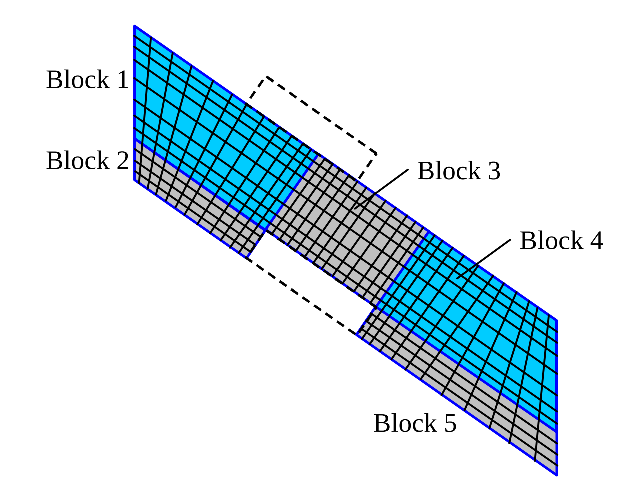



**FIGURE 1**
The blocking topology and a coarse structured grid for a periodic computational domain. This blocking topology applies to CFD codes that allow a block's edges to be split for application of boundary conditions, and do *not* require periodic edge pairs to have identical node distributions.

Unfortunately, many CFD codes require that the node distribution on periodic pairs of edges be identical (the two edges of a periodic pair are "linked" in the grid generation process). In such a case, the grid of Fig. 1 would not be acceptable. Furthermore, although the edges of the blocks of Fig. 1 are not split with respect to adjacent blocks, the top edges of Block 1 and Block 3 are split with respect to the boundary conditions (part of the edge is periodic and part is a wall). Thus these blocks are not really elementary blocks after all. We construct a more elaborate blocking topology in Fig. 2 to correct these problems. The node distribution on the edges of each periodic pair are identical, at the expense of more complexity (7 instead of 5 blocks) and more cell skewness.

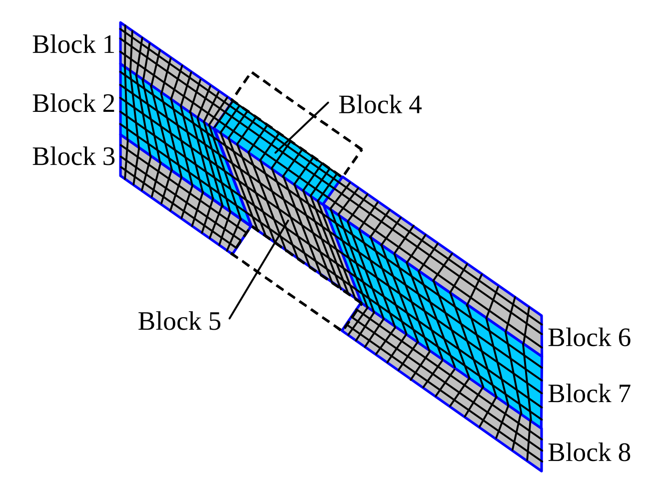



**FIGURE 2**
The blocking topology and a coarse structured grid for a periodic computational domain. This blocking topology applies to CFD codes that require elementary blocks and require periodic edge pairs to have identical node distributions.

***Discussion*** Some of the cells have moderate skewness with the blocking topology of Fig. 2, especially near the corners of Block 2 and Block 7 and throughout Block 5. A more complicated topology can be devised to reduce the amount of skewness.

## 15-23

**Solution** We are to discuss why there is reverse flow in this CFD calculation, and then we are to explain what can be done to correct the problem.

***Analysis*** **Reverse flow at an outlet is usually an indication that the computational domain is not large enough**. In this case the rectangular heat exchanger tubes are inclined at 35$^o$, and the flow will most certainly separate, leaving large recirculating eddies in the wakes. **Anita should extend the computational domain in the horizontal direction downstream** so that the eddies have a chance to "close" and the flow has a chance to re-develop into a flow without any reverse flow.

***Discussion*** In most commercial CFD codes a warning will pop up on the computer monitor whenever there is reverse flow at an outlet. This is usually an indication that the computational domain should be enlarged.

## 15-24

**Solution** We are to generate a computational domain and label all appropriate boundary conditions for two stages of a heat exchanger.

***Analysis*** We look for the smallest computational domain that takes advantage of the periodicity of the geometry. There are several ways to create a periodic grid for this flow. The simplest computational domain consists of a single flow passage between two neighboring tubes. We can make the periodic edge intercept anywhere on the fore and aft portions of the heat exchanger that we desire. We choose the periodic computational domain sketched in Fig. 1.

Boundary conditions are also straightforward, and are labeled in Fig. 1. For a known inlet velocity we set the boundary condition at the left edge as a **velocity inlet**. The tube walls are obviously set as **walls**. The outlet can be set as either a **pressure outlet** or an **outflow**, depending on the provided information and how far the outlet region extends beyond the tubes. Finally, we set three pairs of **translationally periodic** boundaries, one fore, one mid, and one aft of the tubes. We label them separately to avoid confusion.

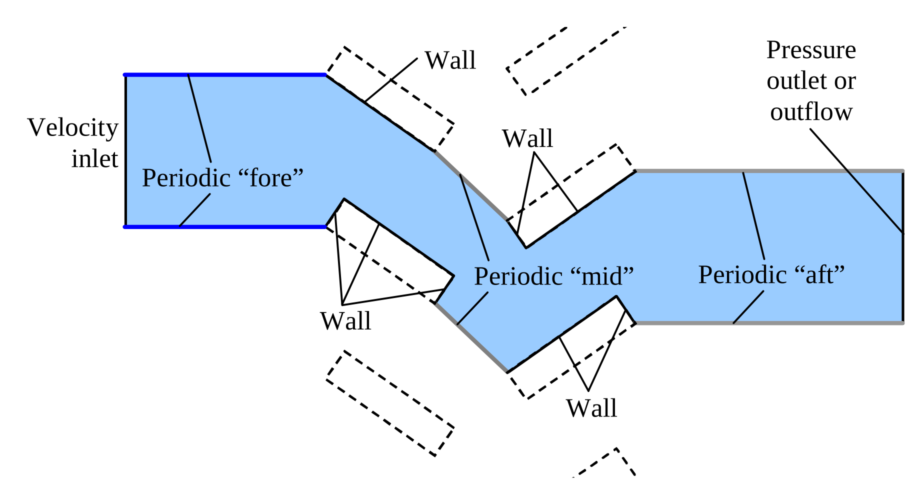



**FIGURE 1**
A periodic computational domain for a given geometry. Boundary conditions are also labeled.

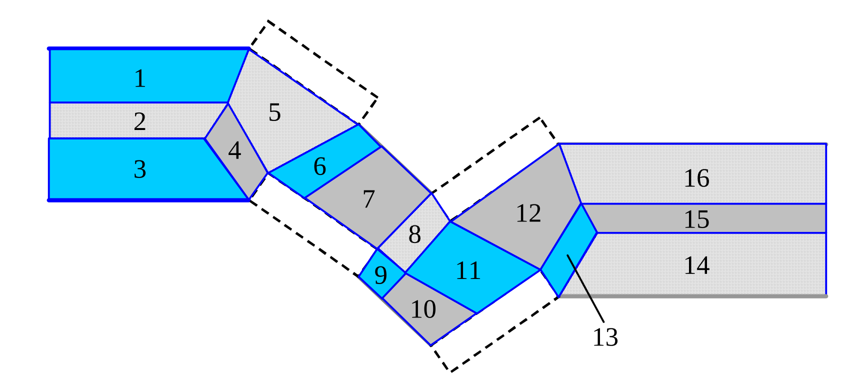
***Discussion*** Many other equally acceptable computational domains are possible.

**15-25**

**Solution** We are to sketch a structured multiblock grid with four-sided elementary blocks for a given computational domain.

***Analysis*** We choose the computational domain of Fig. 1 of the previous problem. We sketch one possible elementary blocking topology in Fig. 1 for the case in which the CFD code requires the node distribution to be exactly the same on periodic pairs. We also assume that we cannot split one periodic edge and not its partner. The blocks are numbered. This topology has **16 elementary blocks**.

**FIGURE 1**
The elementary blocking topology for a periodic computational domain. This blocking topology applies to CFD codes that do not allow a block's edges to be split for application of boundary conditions, and requires periodic edge pairs to have identical node distributions.




Note that with this blocking topology we had to split the periodic "mid" boundary pair into two edges (the tops of blocks 6 and 7 and the bottoms of blocks 9 and 10). As long as both pairs of each segment are the same size and have the same number of nodes, this is not a problem. In the CFD code we would have to name each periodic pair separately, however. The block numbers are labeled. Notice that most of the blocks are nearly rectangular such that none of the computational cells would have to be highly skewed.

***Discussion*** This seems like a rather complicated blocking topology. It would require a bit of work to generate the grid. However, the time spent on developing a good grid is usually well worth the effort. By reducing the amount of cell skewness, we are able to speed up the CFD calculations and obtain more accurate results. This kind of topology also enables us to cluster cells near walls and wakes as needed.

## FlowLab Problems

### 15-26

**Solution** We are to generate CFD solutions for external flow over a 2-D block. Specifically, we are to compare drag coefficient for various values of $R/D$ (the extent of the outer boundary of the computational domain). In addition, we are to compare the calculated value of $C_D$ with experiment.

***Assumptions*** **1** The flow is two-dimensional and incompressible. **2** The flow is symmetric about the $x$ axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is air with $\rho$ = 1.225 kg/m$^3$ and $\mu$ = 1.7894 × 10$^{-5}$ kg/m·s.

**TABLE 1**
Drag coefficient as a function of the normalized extent of the computational domain for turbulent flow over a rectangular block.

| $R/D$ | $C_D$ |
|---|---|
| 5 | 1.81927 |
| 10 | 1.50662 |
| 20 | 1.41076 |
| 50 | 1.36723 |
| 100 | 1.35282 |
| 200 | 1.34671 |
| 500 | 1.34408 |

***Analysis***
($a$) The Reynolds number is

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{\left(1.225\text{ kg/m}^3\right)\left(2.0\text{ m/s}\right)\left(0.10\text{ m}\right)}{1.7894\times10^{-5}\text{ kg/m}\cdot\text{s}} = \mathbf{1.37\times10^4}$$

Experimental data indicate that the drag coefficient for this body is $\boldsymbol{C_D \approx 1.9}$ at Reynolds numbers greater than 10$^4$ (see Chap. 11).

($b$) The CFD code is run for eight values of $R$, all else being equal. $C_D$ is tabulated as a function of $R/D$ in Table 1, and plotted in Fig. 2. As the extent of the computational domain grows in size, the drag coefficient decreases steadily, but levels off to three significant digits of precision by $R/D \approx 200$. Thus, a computational domain extent of **R/D ≈ 100** is sufficient to achieve independence of $C_D$. We report a final value of $\boldsymbol{C_D = 1.34}$.

($c$) There are several possible reasons for the discrepancy between the calculated value of $C_D$ (1.34) and the experimentally obtained value of $C_D$ (about 1.9). First of all, the actual flow is most likely unsteady, with vortices being shed into the wake, whereas we are simulating a steady flow. In addition, the unsteady shedding of vortices renders the flow no longer symmetric about the $x$ axis, whereas we are forcing our flow to be symmetric. Furthermore, the grid resolution may not be adequate to achieve grid independence (this is checked in the following problem). Finally (and most importantly), we are using a *turbulence model* to simulate this flow field. The CFD solution we obtain is only as good as the degree to which the turbulence model correctly models the physics of the turbulence. As discussed in the text, no turbulence model is universally valid for all types of turbulent flows. Discrepancies between experiment and CFD will always exist regardless of how fine the grid or how large the extent of the computational domain.

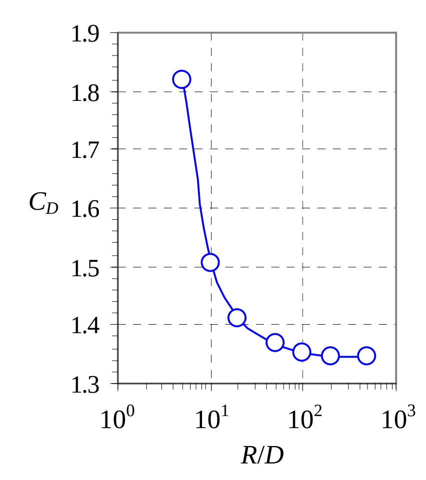



**FIGURE 1**
Drag coefficient plotted as a function of the normalized extent of the computational domain for turbulent flow over a rectangular block.

($d$) Streamlines near the body are plotted for $R/D$ = 5 and 500 in Fig. 2. We notice that the streamlines for the $R/D$ = 5 case are more tight around the body compared to those for $R/D$ = 500. This is most likely due to interference from the outer edges of the computational domain, which are too close for the $R/D$ = 5 case.

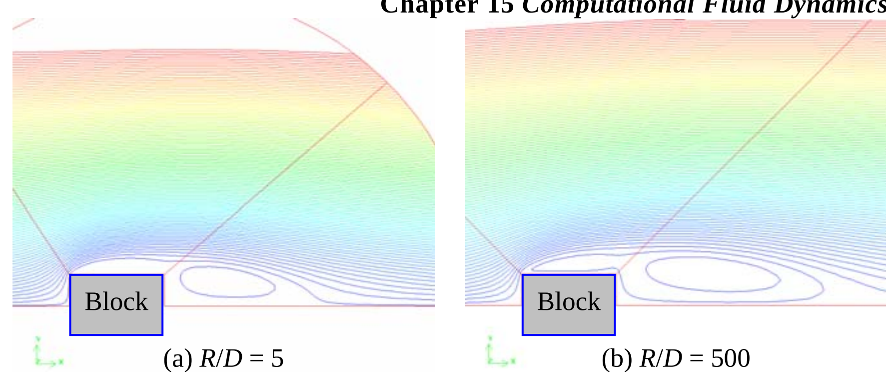



**FIGURE 2**
Streamlines for steady, incompressible, two-dimensional, turbulent flow over a rectangular block at $R/D$ = (a) 5, (b) 500. Only the upper half of the flow is simulated

***Discussion*** Newer versions of FlowLab may give slightly different results. It is perhaps surprising how far away the edges of the computational domain must be in order for them not to influence the flow field around the body. No attempt was made here to optimize the grid, and although the comparison between various grid extents is valid, the solution itself may not be grid independent; the actual value of drag coefficient may not be correct due to lack of grid resolution. Grid independence is analyzed in the next problem.

---

## 15-27

**Solution** We are to test for grid independence by running various values of grid resolution, and we are to examine the effect of grid resolution on drag coefficient.

***Assumptions*** **1** The flow is two-dimensional and incompressible. **2** The flow is symmetric about the $x$ axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is air with $\rho$ = 1.225 kg/m$^3$ and $\mu$ = 1.7894 × 10$^{-5}$ kg/m·s.

***Analysis*** We run the CFD code for several grid resolutions, and we list drag coefficient as a function of total number of computational cells in Table 1. Although some discrepancies exist in the third or fourth digit, the drag coefficient levels off to three significant digits of precision by the sixth row of the table. Further grid refinement may be necessary. We report a final value of $\boldsymbol{C_D}$ **= 1.33** (to three significant digits of precision for comparison with experimental results). This result differs from the experimentally obtained value of $C_D$ = 1.9 by about 30%. We have achieved grid independence to three digits, and thereby eliminate lack of grid resolution as a source of the discrepancy. The other reasons for the discrepancy between CFD and experiment remain, regardless of how fine the grid. Namely, nonuniversality of the turbulence model, unsteadiness, and nonsymmetry.

**TABLE 1**
Drag coefficient as a function of number of cells in the computational domain for turbulent flow over a rectangular block.

| Number of cells | $C_D$ |
|---|---|
| 3120 | 1.45505 |
| 10400 | 1.35249 |
| 12480 | 1.34893 |
| 18720 | 1.33839 |
| 23920 | 1.33595 |
| 26000 | 1.33379 |

***Discussion*** Newer versions of FlowLab may give slightly different results. This exercise illustrates that grid refinement does not necessarily lead to improved CFD predictions.

---

## 15-28

**Solution** We are to repeat the CFD calculation of drag coefficient around a rectangular block for two other fluids – water and kerosene, and we are to compare our results to those obtained using air as the fluid.

***Assumptions*** **1** The flow is two-dimensional and incompressible. **2** The flow is symmetric about the *x* axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The density and viscosity of the default air are $\rho$ = 1.225 kg/m$^3$ and $\mu$ = 1.7894 × 10$^{-5}$ kg/m·s. The density and viscosity of liquid water at $T$ = 15$^o$C are 998.2 kg/m$^3$ and 1.003 × 10$^{-3}$ kg/m·s. The density and viscosity of kerosene at $T$ = 15$^o$C are 780.0 kg/m$^3$ and 2.40 × 10$^{-3}$ kg/m·s.

**TABLE 1**
Drag coefficient as a function of fluid type for the case of turbulent flow over a rectangular block. In all cases, the Reynolds number is the same.

| Fluid | Re | $C_D$ |
|---|---|---|
| Air | $1.37 \times 10^4$ | 1.34344 |
| Water | $1.37 \times 10^4$ | 1.34343 |
| Kerosene | $1.37 \times 10^4$ | 1.34343 |

***Analysis*** A comparison of the CFD calculations for all three fluids is given in Table 1. The drag coefficient is identical to about five digits of precision. We conclude that for incompressible flow without free surface effects, the Reynolds number is the critical parameter; **the type of fluid is irrelevant provided that the Reynolds number is the same**. This reinforces what we learned about dimensional analysis in Chap. 7.

***Discussion*** Newer versions of FlowLab may give slightly different results. Some incompressible CFD codes work with normalized variables from the start, requiring input of a Reynolds number instead of dimensional quantities such as velocity, density, and viscosity.

## 15-29

**Solution** We are to generate CFD solutions for drag coefficient as a function of Reynolds number and compare and discuss.

***Assumptions*** **1** The flow is two-dimensional and incompressible. **2** The flow is symmetric about the *x* axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is air with $\rho$ = 1.225 kg/m$^3$ and $\mu$ = 1.7894 × 10$^{-5}$ kg/m·s.

**TABLE 1**
Drag coefficient as a function of Reynolds number for turbulent flow over a rectangular block.

| Re | $C_D$ |
|---|---|
| 10000 | 1.30788 |
| 50000 | 1.40848 |
| 100000 | 1.42215 |
| 500000 | 1.43065 |
| 1.00E+06 | 1.43194 |
| 3.00E+06 | 1.43296 |
| 5.00E+06 | 1.43431 |

***Analysis*** We compare six cases in Table 1. We see that $C_D$ levels off to a value of 1.43 to three digits of precision for Re greater than about 5 × 10$^5$. Thus, **we have achieved Reynolds number independence**, although the required Reynolds number is somewhat larger than that required experimentally.

The last two cases are peculiar in that the Mach numbers are well beyond the incompressible limit (around 0.3) since the speed of sound in air at room temperature is around 340 m/s. However, even though these flows are unphysical, the CFD code is run as incompressible, and is not “aware” of this problem since the speed of sound is treated as infinite in an incompressible flow solver. The comparison with Re is still valid since we can use *any* incompressible fluid for the calculations, as illustrated in the previous problem.

***Discussion*** Newer versions of FlowLab may give slightly different results. Reynolds number independence checks are not always as simple as that shown here, because as Re increases, boundary layer thicknesses tend to decrease, requiring a finer mesh near walls. In the present problem this is not really an issue because the flow separates at the sharp edges of the block, and boundary layer thickness is not an important parameter in calculation of the drag.

**15-30**

**Solution** We are to generate CFD solutions using several turbulence models, and we are to compare and discuss the results, particularly the drag coefficient.

***Assumptions*** **1** The flow is two-dimensional and incompressible. **2** The flow is symmetric about the $x$ axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is air with $\rho$ = 1.225 kg/m$^3$ and $\mu$ = 1.7894 × 10$^{-5}$ kg/m·s.

***Analysis*** All cases are run at the same Reynolds number, namely 1.37 × 10$^4$. We compare drag coefficient for all four cases in Table 1. We see that $C_D$ depends greatly on turbulence model. Most of the models underpredict the drag coefficient by about 30%, but the $k$-$\omega$ model overpredicts $C_D$ by more than 33%. In terms of percentage error, the Reynolds stress model is closest to the experimental value of 1.9.

**TABLE 1**

Drag coefficient as a function of turbulence model for flow over a rectangular block. The error is in comparison to the experimental value of 1.9.

| Turbulence model | $C_D$ | Error (%) |
|---|---|---|
| $k$-$\varepsilon$ (2 eq.) | 1.34342 | -29.5% |
| $k$-$\omega$ (2 eq.) | 2.536939 | 33.5% |
| Spallart-Allmaras (1 eq.) | 1.360602 | -28.4% |
| Reynolds stress model (5 eq.) | 1.385374 | -27.1% |

***Discussion*** Newer versions of FlowLab may give slightly different results. Turbulence models involve semi-empirical analysis, curve-fits, and simplifications, and **no turbulence model is best for every kind of fluid flow**. It is not always clear which turbulence model to use for a given problem.

**15-31**

**Solution** We are to run CFD simulations of flow over a 2-D rectangular block with various values of $L/D$. We are to compare and discuss streamlines and drag coefficient, and we are to compare our CFD results to experiment.

***Assumptions*** **1** The flow is two-dimensional and incompressible. **2** The flow is symmetric about the $x$ axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is air with $\rho$ = 1.225 kg/m$^3$ and $\mu$ = 1.7894 × 10$^{-5}$ kg/m·s.

***Analysis*** (*a*) Drag coefficient is tabulated as a function of $L/D$ in Table 1. These data are also plotted in Fig. 1, along with experimental data from Chap. 11. We see that the calculations are consistently lower than the experimental values for the smaller lengths, but the agreement is very good at $L/D$ = 3 (as high as the table goes). While the experimentally obtained drag coefficient peaks at $L/D$ = 0.5, the CFD calculations predict that $C_D$ decays continually with $L/D$.

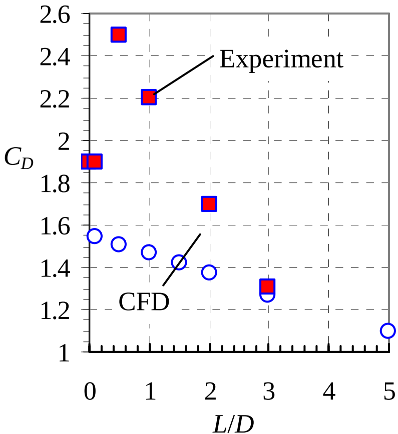



**FIGURE 1**
Drag coefficient as a function of $L/D$ for turbulent flow over a rectangular block.

(*b*) For several values of $L/D$, we plot streamlines near the block (Fig. 2). We can see why the drag coefficient decreases as $L/D$ increases. Namely, as the block length increases, the large recirculating flow regions in the wake (wake eddies) decrease in size (particularly in the vertical direction). In all cases, the flow separates at the sharp corner of the blunt face, but as $L$ increases, the flow has more time to flatten out along the upper and lower walls of the block, leading to wake eddies of reduced thickness. Indeed, by $L/D$ = 3.0 (Fig. 2f) the separated flow along the upper and lower walls appears to reattach just upstream of the back of the block. The wake eddies in this case are much thinner than those of the shorter blocks (compare Fig. 2b and 2e for example). Another way to express this is to say that the longer blocks are more "streamlined" (in a gross sense of the word) than are the shorter blocks, and therefore have less drag. Experiments show that the drag is highest at $L/D$ = 0.5. Apparently the vortex shedding process is strong for this case, leading to high drag. Our steady, symmetric CFD model is not able to simulate the unsteady features of the actual flow.

**TABLE 1**
Drag coefficient as a function of $L/D$ for turbulent flow over a rectangular block.

| $L/D$ | $C_D$ |
|---|---|
| 0.1 | 1.53997 |
| 0.5 | 1.50858 |
| 1 | 1.46466 |
| 1.5 | 1.42077 |
| 2 | 1.36825 |
| 3 | 1.26405 |
| 5 | 1.09386 |

(*c*) There are many possible reasons for the discrepancy between CFD calculations and experiment. We are modeling the problem as a steady flow that is symmetric about the axis, but experiments reveal that flow over bluff bodies like these oscillate and shed vortices – **the flow is neither steady nor symmetric**. Furthermore, we are using a turbulence model. As discussed previously, **turbulence models are *not* universal**, and may not be applicable to the present problem. **A DNS or LES simulation would be required to correctly model the unsteady turbulent eddies**.

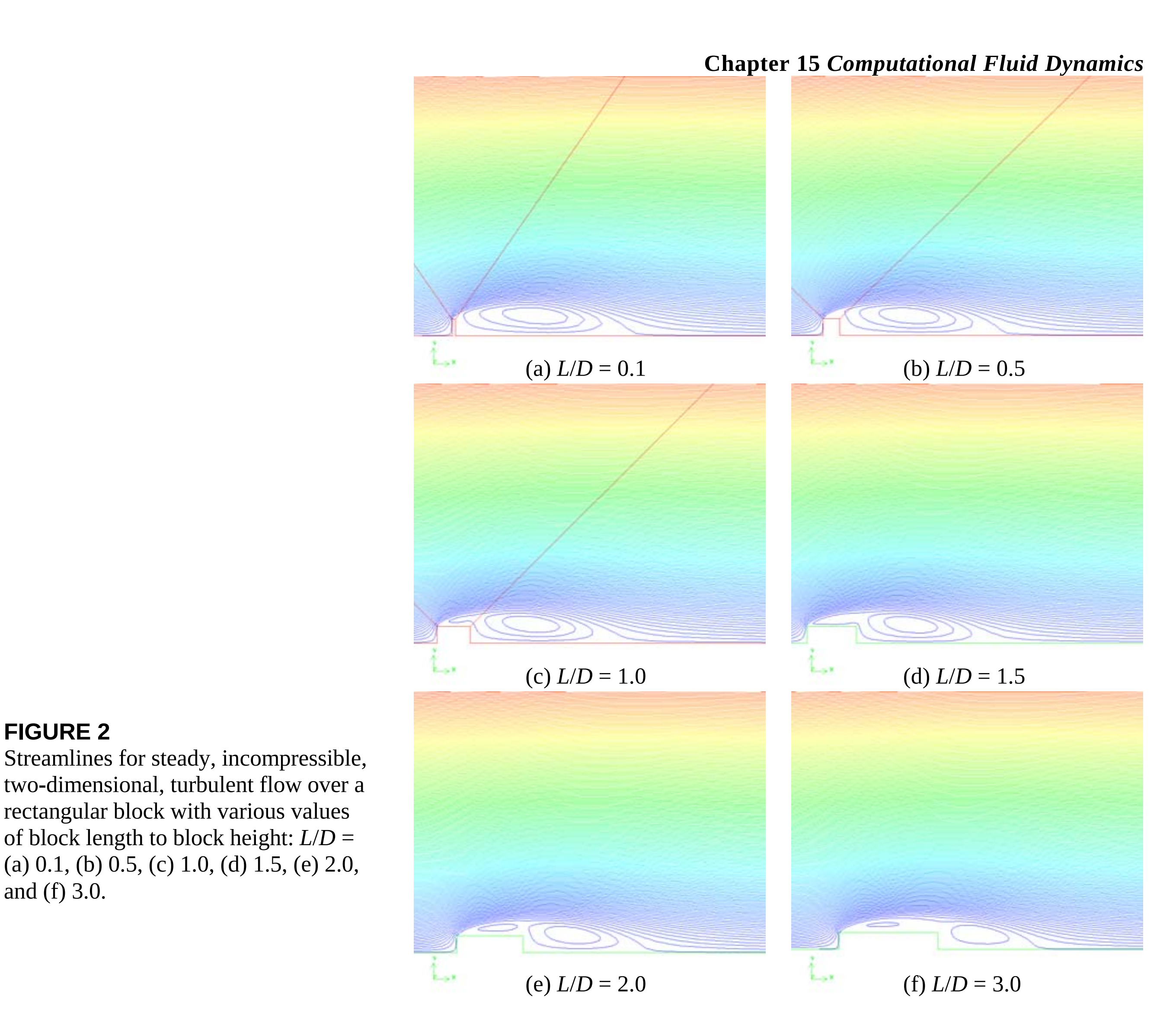



**FIGURE 2**
Streamlines for steady, incompressible, two-dimensional, turbulent flow over a rectangular block with various values of block length to block height: $L/D$ = (a) 0.1, (b) 0.5, (c) 1.0, (d) 1.5, (e) 2.0, and (f) 3.0.

***Discussion*** Newer versions of FlowLab may give slightly different results. CFD issues such as grid resolution and the extent of the computational domain do not contribute appreciably to the discrepancy here, since we have set up the computational domain based on results of similar previous problems. As $L/D$ increases, the discrepancy between CFD results and experiment gets smaller. This can be explained by the fact that the shed vortices are reduced in strength as $L/D$ increases – the steady, symmetric CFD simulation thus becomes more physically correct with increasing $L/D$.

## 15-32

**Solution** We are to generate CFD solutions for external flow over a cylindrical block. Specifically, we are to compare drag coefficient for various values of $R/D$ (the extent of the outer boundary of the computational domain). In addition, we are to compare the calculated value of $C_D$ with experiment.

***Assumptions*** **1** The flow is incompressible. **2** The flow is axisymmetric about the $x$ axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is air with $\rho = 1.225$ kg/m$^3$ and $\mu = 1.7894 \times 10^{-5}$ kg/m·s.

**TABLE 1**
Drag coefficient as a function of the normalized extent of the computational domain for turbulent flow over a cylindrical block.

| $R/D$ | $C_D$ |
|---|---|
| 5 | 1.0704 |
| 10 | 1.04053 |
| 20 | 1.03567 |
| 50 | 1.02155 |
| 100 | 1.00781 |
| 200 | 0.998819 |
| 500 | 0.992311 |

***Analysis***
(*a*) The Reynolds number based on cylinder diameter is the same as that of Problem 15-26, namely,

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{\left(1.225 \text{ kg/m}^3\right)\left(2.0 \text{ m/s}\right)\left(0.10 \text{ m}\right)}{1.7894\times10^{-5} \text{ kg/m}\cdot\text{s}} = \mathbf{1.37\times10^4}$$

Experimental data indicate that the drag coefficient for this body is $C_D \approx$ **0.90** at Reynolds numbers greater than $10^4$ (see Chap. 11).

(*b*) The CFD code is run for eight values of $R$, all else being equal. $C_D$ is tabulated as a function of $R/D$ in Table 1, and plotted in Fig. 2. As the extent of the computational domain grows in size, the drag coefficient decreases steadily, but is trying to level off by $R/D \approx 500$. Thus, a computational domain extent of **R/D ≈ 500 or more** is needed to achieve independence of $C_D$. We report a final value of $C_D$ **= 0.99**. Unfortunately, the program does not allow for $R/D$ values greater than 500.

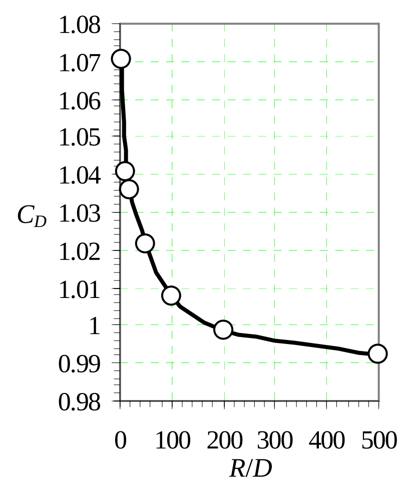



**FIGURE 1**
Drag coefficient plotted as a function of the normalized extent of the computational domain for turbulent flow over a rectangular block.

(*c*) There are several possible reasons for the discrepancy between the calculated value of $C_D$ and the experimentally obtained value of $C_D$ (about 0.90). First of all, the actual flow is most likely unsteady, with vortices being shed into the wake, whereas we are simulating a steady flow. In addition, the unsteady shedding of vortices renders the flow no longer axisymmetric about the $x$ axis, whereas we are forcing our flow to be axisymmetric. Furthermore, the grid resolution may not be adequate to achieve grid independence. Finally (and most importantly), we are using a *turbulence model* to simulate this flow field. The CFD solution we obtain is only as good as the degree to which the turbulence model correctly models the physics of the turbulence. As discussed in the text, no turbulence model is universally valid for all types of turbulent flows. Discrepancies between experiment and CFD will always exist regardless of how fine the grid or how large the extent of the computational domain.

(*d*) Streamlines near the body are plotted for $R/D$ = 5 and 500 in Fig. 2. There is surprisingly little observable difference between these two extreme cases. Compared to the 2-D case, it appears that **the axisymmetric case requires a greater computational domain extent**. This is surprising, because in an axisymmetric flow field, the fluid can flow around the body in all directions, not just over the top and bottom as in 2-D flow. As fluid moves away from the body, the radius also increases there, and more mass can flow through that radial location compared to the 2-D case. In other words an axisymmetric flow is less "confined" or "constrained" than a corresponding two-dimensional flow. Thus, we might have expected the opposite behavior.

We compare our predicted drag coefficient (0.99) with that of experiment (0.90). The discrepancy is only about 10% – much better agreement than the two-dimensional case. The reasons for this improvement is not clear. Axisymmetric flows tend to be less unsteady, and the vortices they shed are generally less coherent and weaker. This may contribute to some of the improvement. It may be merely fortuitous that the turbulence model yields better results for the axisymmetric case as compared to the 2-D case.

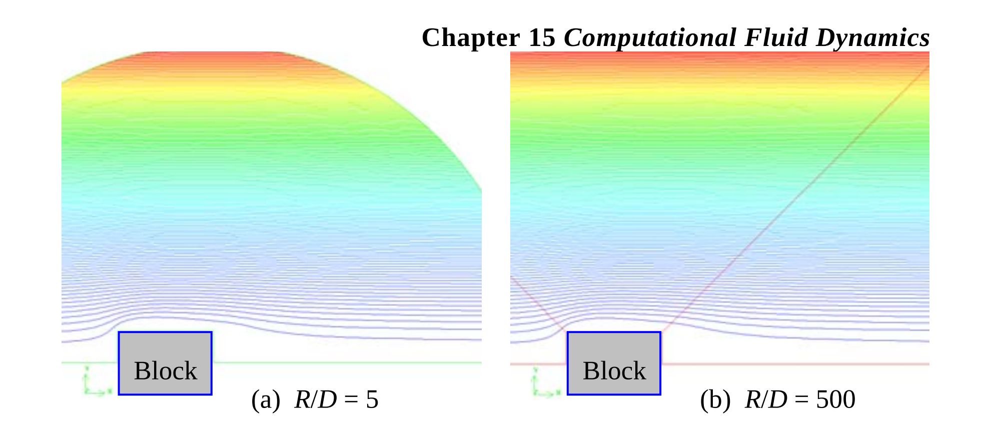



**FIGURE 2**
Streamlines for steady, incompressible, axisymmetric, turbulent flow over a rectangular block at $R/D$ = (a) 5, (b) 500. Only one slice of the flow is shown.

***Discussion*** Newer versions of FlowLab may give slightly different results. The far field boundaries must be quite far away to achieve results that are independent of the extent of the boundary.

**15-33**

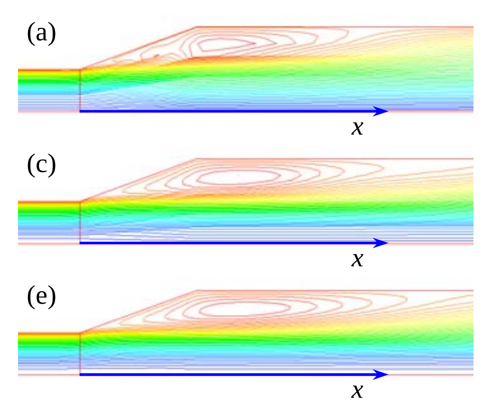
**Solution** We are to generate CFD solutions for several grid resolutions to test for grid independence for flow through a diffuser. Specifically, we are to compare streamlines and pressure difference at each level of grid resolution.

***Assumptions*** **1** The flow is incompressible. **2** The flow is axisymmetric about the *x* axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is air with $\rho = 1.225$ kg/m$^3$ and $\mu = 1.7894 \times 10^{-5}$ kg/m·s.

***Analysis***

(*a*) Streamlines are plotted for six grid resolution cases in Fig. 1.



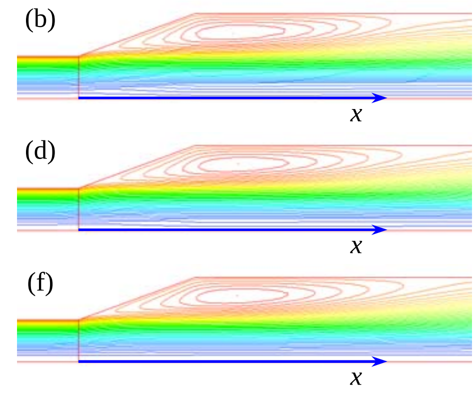




**FIGURE 1**
Streamlines for steady, incompressible, axisymmetric, turbulent flow through a diffuser at various levels of grid resolution; the number of cells is (a) 880, (b) 3520, (c) 4950, (d) 8800, (e) 13750, and (f) 19800.

At the very coarse grid resolution, the streamlines are not well defined (Fig. 1a), and the calculation is not reliable. As grid resolution improves, details of the flow separation region become more refined – the boundary layer is unable to remain attached in such a strong adverse pressure gradient. From these plots, it appears that grid independence has been achieved by about the fourth case (Fig. 1d), beyond which there is no noticeable change in the shape of the streamlines. We note that simulation of flow separation and separation bubbles is often a very difficult task for a CFD program. In this particular problem we must use an extremely fine grid in order to resolve the details of the flow separation.

(*b*) $\Delta P$ is tabulated as a function of cell count in the table for the case with $\theta = 20^{\circ}$. As grid resolution improves, $\Delta P$ increases, and becomes independent of grid resolution by the fourth or fifth mesh. This is also seen in Fig. 2 where $\Delta P$ is plotted as a function of cell count.

| cell count | $\Delta P$ (Pa) |
|---|---|
| 880 | -34.5622 |
| 3520 | -32.7267 |
| 4950 | -32.4183 |
| 8800 | -32.2021 |
| 13750 | -32.1162 |
| 19800 | -32.1099 |

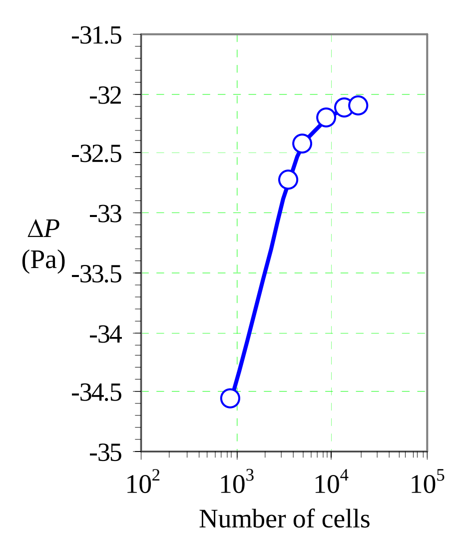



**FIGURE 2**
Pressure difference as a function of the number of cells in the computational domain. Turbulent flow through an axisymmetric diffuser.

***Discussion*** The unphysical-looking streamlines at the very coarse grid resolution (Fig. 1a) are due to interpolation errors when the CFD code calculates contours of constant stream function. Notice that even though there is gross flow separation in this diffuser, there is still a pressure recovery through the diffuser ($P_{in}$ is less than $P_{out}$). A better design (with even higher pressure recovery) would use a smaller diffuser angle so as to avoid flow separation along the diffuser wall. The outlet of the computational domain is reasonably far downstream (several pipe diameters) to avoid reverse flow at the outlet.

**15-34**

**Solution** We are to compare the pressure drop and the pressure distribution at the outlet of a diffuser in a round pipe with two different outlet conditions: pressure outlet and outflow.

***Assumptions*** **1** The flow is incompressible. **2** The flow is axisymmetric about the $x$ axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is air with $\rho$ = 1.225 kg/m$^3$ and $\mu$ = 1.7894 × 10$^{-5}$ kg/m·s.

***Analysis*** For the present case (outflow boundary condition), the pressure difference is $\Delta P = P_{in} - P_{out}$ = **–32.1096 Pa**, which agrees to four digits of precision with the result of the previous problem at the same grid resolution, for which $\Delta P = P_{in} - P_{out}$ = **–32.1090 Pa**. Thus we conclude that **the outlet boundary condition has negligible effect on this flow field**.

For the case with the pressure outlet boundary condition, the static pressure at the outlet of the computational domain is forced to be constant (zero gage pressure in the calculations of the previous problem). In the present case however, the outflow boundary condition does *not* fix static pressure – rather, it forces flow variables to level off as they approach the outlet boundary. We find that $P$ varies by less than 0. percent across the boundary. Thus, even though $P$ is not forced to be constant along the outflow boundary, it turns out to be nearly constant anyway.

***Discussion*** Newer versions of FlowLab may give slightly different results. With a velocity inlet and an outflow outlet, we do not fix the value of pressure at either boundary. Instead, the CFD code assigns $P$ = 0 gage pressure at some (arbitrary) location in the flow field (the default location is the origin). Even though the inlet and outlet pressures differ significantly between the two cases, $\Delta P$ is identical to within four significant digits of precision. These results verify the statement we made in Chap. 9: For incompressible flow, it is not pressure itself which is important to the flow field, but rather pressure *differences*. We conclude that the differences between pressure outlet and outflow are small, provided that the outlet boundary is far enough away from the region of interest in the flow field.

**15-35**

**Solution** We are to generate CFD solutions for flow through a diffuser at various values of diffuser half-angle $\theta$. Specifically, we are to compare streamlines and pressure difference for each case, and we are to determine the maximum value of $\theta$ that achieves the stated design objectives.

***Assumptions*** **1** The flow is incompressible. **2** The flow is axisymmetric about the $x$ axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is air with $\rho = 1.225$ kg/m$^3$ and $\mu = 1.7894 \times 10^{-5}$ kg/m·s.

***Analysis***
($a$) Streamlines are plotted in Fig. 1 for all twelve cases.

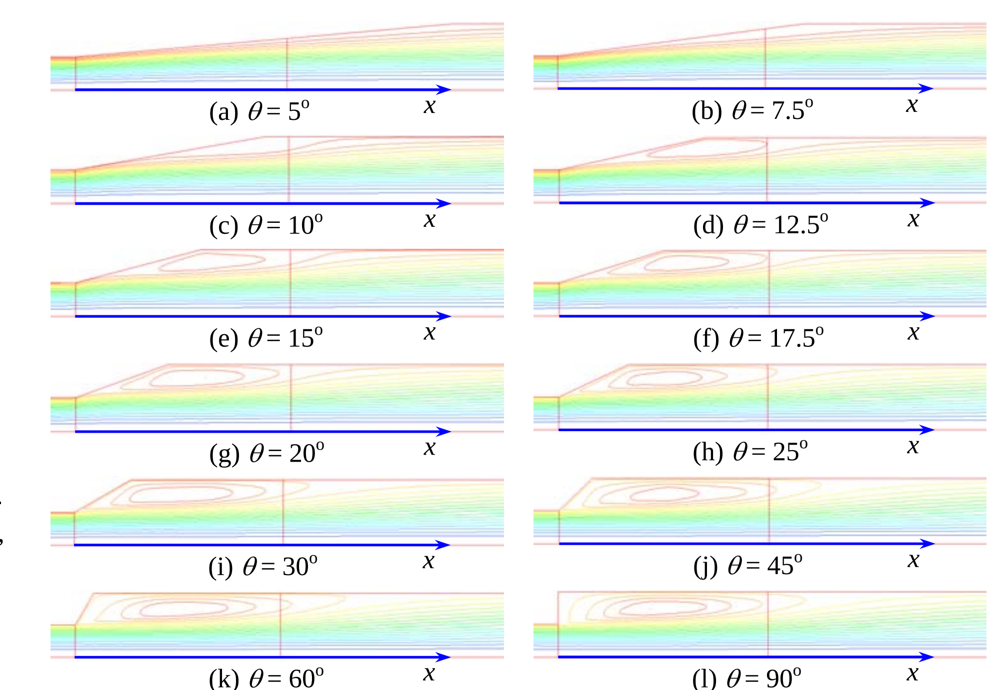



**FIGURE 1**
Streamlines for steady, incompressible, axisymmetric, turbulent flow through a diffuser at various values of diffuser half-angle: $\theta$ = (a) 5$^o$, (b) 7.5$^o$, (c) 10$^o$, (d) 12.5$^o$, (e) 15$^o$, (f) 17.5$^o$, (g) 20$^o$, (h) 25$^o$, (i) 30$^o$, (j) 45$^o$, (k) 60$^o$, and (l) 90$^o$. (The latter case is a sudden expansion.)

At $\theta$ = 5$^o$ and 7.5$^o$, the flow does not separate along the diffuser wall (Figs. 1a and 1b), although separation appears imminent near the downstream corner of the diffuser for the latter case. **To avoid flow separation, Barb should recommend a diffuser half-angle of 7.5$^o$ or less**. As $\theta$ increases, the boundary layer is unable to remain attached in the adverse pressure gradient, and the flow separates. A very small separation bubble is apparent at $\theta$ = 10$^o$ (Fig. 1c). As $\theta$ continues to increase, the separation bubble grows in size, and the separation point moves upstream, closer and closer to the upstream corner of the diffuser (compare Figs. 1d through 1g). By $\theta$ = 25$^o$ (Fig. 1h), the flow separates very close to the start of the diffuser. From this point on, the diffuser angle is so sharp that the flow separates right at the upstream corner of the diffuser. The streamline patterns reveal that the separation bubble continues to grow in size as $\theta$ increases (Figs. 1i through 1j). Beyond $\theta \approx 45^o$ however, the streamline pattern changes very little in the separation bubble (compare Figs. 1k and 1l).

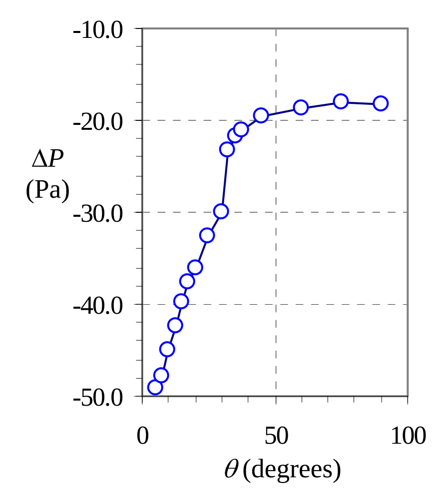



**FIGURE 2**
Pressure difference from inlet to outlet as a function of diffuser half-angle $\theta$.

(*b*) $\Delta P$ is tabulated as a function of diffuser half-angle in the table (four extra cases are solved for improved clarity). As $\theta$ increases, $P_{in}$ increases, reflecting the effect of the larger separation bubble. Physically, we achieve less and less pressure recovery as the separation bubble grows. We see that $\Delta P$ flattens out at high values of $\theta$, becoming nearly independent of $\theta$ beyond $\theta \approx 60^o$. This is also seen in Fig. 2 where $P_{in}$ is plotted as a function of diffuser half-angle. There is a sharp rise in $\Delta P$ beyond $30^o$. The reason for this is not certain, but is probably related to the fact that when $\theta$ is greater than about $30^o$, the flow separates right at the upstream corner. The pressure rise is greater than 40 Pa for all angles below $15^o$. Thus, to ensure a pressure recovery of at least 40 pascals, **Barb should recommend a diffuser half-angle of 12.5$^o$ or less**.

| $\theta$ | $\Delta P$ |
|---|---|
| 5 | -49.1371 |
| 7.5 | -47.7787 |
| 10 | -44.9927 |
| 12.5 | -42.4013 |
| 15 | -39.6981 |
| 17.5 | -37.6431 |
| 20 | -36.0981 |
| 25 | -32.7173 |
| 30 | -29.9919 |
| 32.5 | -23.2118 |
| 35 | -21.6434 |
| 37.5 | -21.0490 |
| 45 | -19.6571 |
| 60 | -18.7252 |
| 75 | -18.1364 |
| 90 | -18.3018 |

***Discussion*** The results here are for an older version of FlowLab, and the results at $20^o$ are therefore not exactly the same as those of the previous problem. Newer versions of FlowLab would give slightly different results, and the agreement would be exact. The outlet of the computational domain is reasonably far downstream (several pipe diameters) to avoid reverse flow at the outlet. Notice that even for the case of a sudden expansion ($\theta = 90^o$) there is still a pressure recovery through the diffuser ($P_{in}$ is less than $P_{out}$).

## 15-36

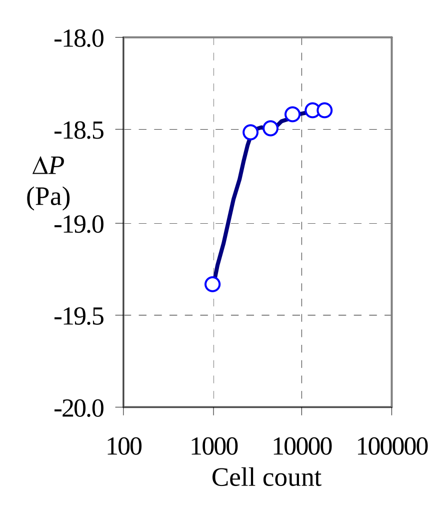
**Solution** We are to perform a grid independence test on the $90^o$ diffuser (sudden expansion) case of the previous problem by refining the grid resolution.

***Assumptions*** **1** The flow is incompressible. **2** The flow is axisymmetric about the *x* axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is air with $\rho$ = 1.225 kg/m$^3$ and $\mu = 1.7894 \times 10^{-5}$ kg/m·s.

***Analysis*** We run several levels of grid refinement, tabulate the results, and plot the results. We have achieved grid independence to the third significant digit of precision by the fourth refinement level, namely, 8100 cells. The final value of $\Delta P$ is reported to three significant digits as **-18.4 Pa**.

| cell count | $\Delta P$ (Pa) |
|---|---|
| 996 | -19.3374 |
| 2700 | -18.5233 |
| 4500 | -18.4956 |
| 8100 | -18.4171 |
| 13500 | -18.3977 |
| 17855 | -18.3984 |




***Discussion*** Newer versions of FlowLab may give slightly different results. We could try refining the grid even further, but the increase in precision would not be worth the effort. In a flow field such as this, the separation point is fixed at the sharp corner; the CFD code does not have problems identifying the separation point. Notice that even for the case of a sudden expansion ($\theta = 90^o$) there is still a pressure recovery through the diffuser ($P_{in}$ is less than $P_{out}$).

**15-37**

**Solution** We are to generate CFD solutions for several downstream tube extension lengths to see the impact of the downstream boundary. We are also to discuss the variation of $P_{in}$ with $L_{extend}$.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is axisymmetric about the $x$ axis. **3** The flow is laminar.

***Properties*** The fluid is water with $\rho$ = 998.2 kg/m$^3$ and $\mu$ = 0.001003 kg/m·s.

***Analysis***
(*a*) Only the first case at $L_{extend}/D_2$ = 0.25 has reverse flow at the pressure outlet. By $L_{extend}/D_2$ = 0.50 the reverse flow problems are gone. The streamlines reveal that the flow separates at the corner, forming a recirculating eddy. The eddy reattaches at approximately $x/D_2$ = 0.50. Streamlines are shown for the first three cases in Fig. 1. The overall streamline shapes appear to be unaffected by the downstream extent of the computational domain.

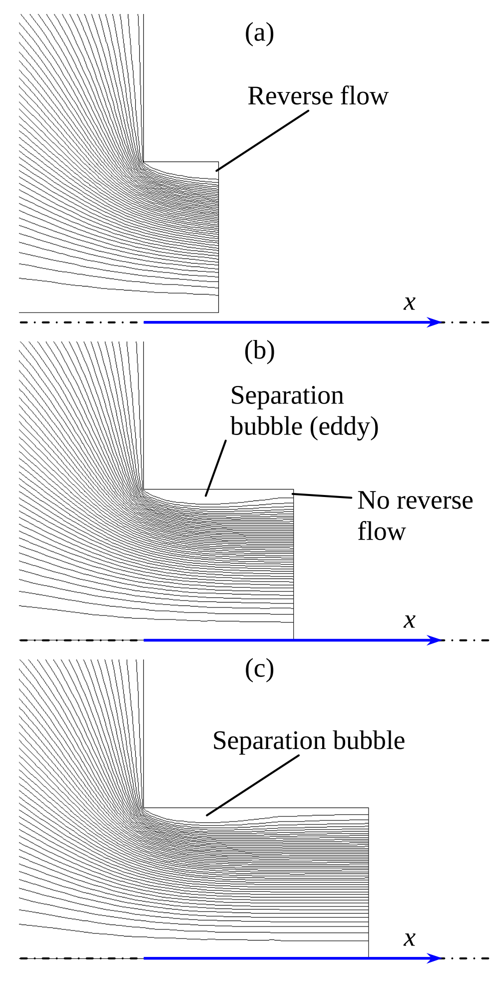



**FIGURE 1**
Streamlines: $L_{extend}/D_2$ = (a) 0.25, (b) 0.50, and (c) 0.75.

(*b*) $P_{in}$, $P_1$, and $\Delta P = P_{in} - P_1$ are tabulated as functions of $L_{extend}/D_2$. $\Delta P$ is independent of $L_{extend}$ (to three significant digits of precision) by $L_{extend}/D_2$ = 0.5.

| $L_{extend}/D_2$ | $P_{in}$ (Pa, gage) | $P_1$ (Pa, gage) | $\Delta P$ (Pa) |
|---|---|---|---|
| 0.25 | 664.924 | 212.156 | 452.767 |
| 0.5 | 622.068 | 171.15 | 450.918 |
| 0.75 | 597.788 | 146.831 | 450.957 |
| 1 | 590.532 | 139.464 | 451.068 |
| 1.25 | 590.036 | 138.86 | 451.176 |
| 1.5 | 592.523 | 141.24 | 451.283 |
| 2 | 601.075 | 149.576 | 451.498 |
| 2.5 | 609.822 | 158.733 | 451.089 |
| 3 | 620.434 | 169.248 | 451.186 |

(*c*) Inlet gage pressure $P_{in}$ is plotted as a function of $L_{extend}/D_2$ in Fig. 2. The inlet pressure drops as $L_{extend}/D_2$ increases from 0.25 to 1.25, but then starts to slowly rise as $L_{extend}/D_2$ continues to increase. We explain this trend with help from the streamlines of Fig. 1. First of all, the data for $L_{extend}/D_2$ = 0.25 are not reliable, since there is reverse flow at the pressure outlet. Thus, we begin our discussion at $L_{extend}/D_2$ = 0.50. The separation bubble (recirculating eddy) at the inlet to the smaller diameter tube forces the flow to converge and accelerate through an effective area that is smaller than the cross-sectional area of the small tube (a *vena contracta*). This high speed flow leads to a low pressure region near the separation bubble. Downstream of the separation bubble, the pressure tries to increase slightly (the downstream portion of the eddy behaves like a diffuser, recovering some of the pressure loss). However, since the outlet pressure is fixed at zero gage pressure, the inlet pressure must decrease to compensate. This explains why $P_{in}$ decreases when $L_{extend}/D_2$ increases from 0.50 to 1.25. By $L_{extend}/D_2$ = 1.25, the outlet of the computational domain is beyond the region of pressure recovery due to the eddy, and the flow begins its slow development towards fully developed laminar pipe flow. Beyond $x/D_2 \approx$ 1.25 the pressure decreases

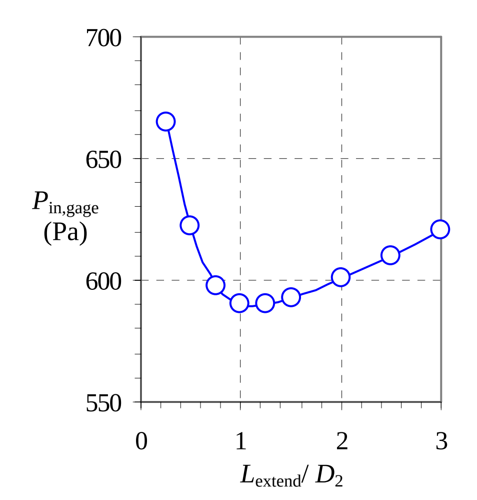



**FIGURE 2**
Inlet gage pressure as a function of normalized downstream extent of the computational domain.

axially along the tube because of friction along the pipe wall. But since the exit pressure is atmospheric, the inlet pressure must rise accordingly. In other words, the pressure at the inlet must rise to overcome the increasing pressure drop through the tube. We expect this trend to continue, eventually becoming linear.

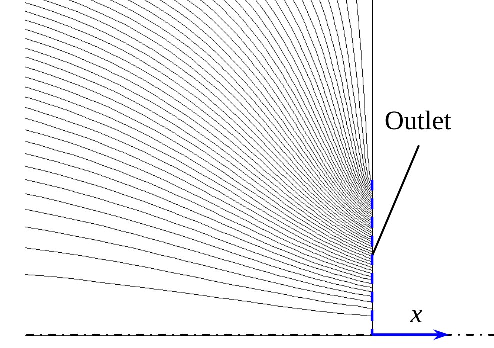
Based on all our results taken collectively, **we recommend that Shane use a value of $L_{extend}/D_2$ of 2.0 or more** to ensure proper simulation of this flow. If computer speed or memory is a problem, he can get away with a minimum of $L_{extend}/D_2$ = 0.75 to avoid reverse flow problems at the outlet.

***Discussion*** For other geometries, fluids, or speeds, the separation bubble may extend farther than in this example. Thus is wise to extend the computational domain reasonably far downstream (several tube diameters).

---

## 15-38

**Solution** We are to simulate flow through a sudden contraction in a tube, using no downstream tube extension. We are to calculate the error in pressure drop and discuss.




***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is axisymmetric about the *x* axis. **3** The flow is laminar.

***Properties*** The fluid is water with $\rho$ = 998.2 kg/m$^3$ and $\mu$ = 0.001003 kg/m·s.

***Analysis*** There is no reverse flow in the CFD simulation. The streamlines (shown in the figure) reveal that since there is no downstream extension, there is obviously no way to have a separation bubble. With the outlet pressure set to a constant value, the streamlines adjust themselves such that the fluid flows through the outlet without reverse flow. Although this may appear to be a good CFD solution, it turns out to be erroneous because the actual pressure across the interface of a sudden contraction is *not* constant. We obtain a net pressure difference of $\Delta P = P_{in} - P_{out}$ = **519.6 Pa**. Compared to the best values from the previous problem ($\Delta P$ = 451 Pa), **the percentage error is about 15%**.

***Discussion*** Newer versions of FlowLab may give slightly different results. The error in $\Delta P$ caused by neglecting the downstream extension is significant. This reinforces our discussion in the text about extending outlets.

---

## 15-39

**Solution** We are to simulate flow through a sudden contraction in a tube, using three different values of outlet pressure at the pressure outlet boundary. We are to compare the pressure drop and discuss.

***Assumptions*** **1** The flow is steady and incompressible. **2** The flow is axisymmetric. **3** The flow is laminar.

***Properties*** The fluid is water with $\rho$ = 998.2 kg/m$^3$ and $\mu$ = 0.001003 kg/m·s.

***Analysis*** Results for the three cases are summarized in the table for the case in which $L_{extend}/D_2$ = 2.0. We see that the inlet pressure and the pressure at *x* = 0 rise or fall in symphony with $P_{out}$, such that the net pressure difference is the *same* (to more than four significant digits) in all cases. The results verify a statement we made in Chap. 9: For incompressible flow, it is not pressure itself which is important to the flow field, but rather pressure *differences*.

| $P_{out}$ (Pa, gage) | $P_{in}$ (Pa, gage) | $P_1$ (Pa, gage) | $\Delta P$ (Pa) |
|---|---|---|---|
| -50,000 | -49,398.9 | -49,850.4 | 451.500 |
| 0 | 601.075 | 149.576 | 451.498 |
| 50,000 | 50,601.1 | 50,149.6 | 451.496 |

***Discussion*** The slight differences (in the fifth digit) in $\Delta P$ are due to numerical inaccuracies in the CFD code (the residuals never go to exactly zero). Note that we are subtracting two large numbers, which inevitably leads to precision errors. If we had run the simulation with double precision arithmetic and adjusted some of the numerical parameters, we could have obtained even better agreement.

---

## 15-40

**Solution** We are to generate CFD solutions for several downstream tube extension lengths to see the impact of the downstream boundary.

***Assumptions*** **1** The flow is incompressible. **2** The flow is axisymmetric about the $x$ axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is water with $\rho$ = 998.2 kg/m$^3$ and $\mu$ = 0.001003 kg/m·s.

***Analysis*** ($a$) Four Reynolds numbers are calculated. Note that since the diameter decreases by a factor of 4 through the contraction, the average velocity in the smaller downstream tube increases by a factor of $4^2$ = 16 compared to that in the larger upstream tube. For the laminar case of Problem 15-37,

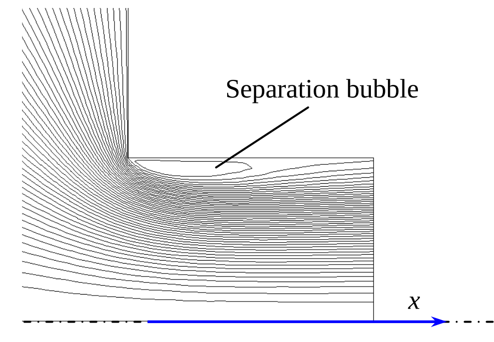



**FIGURE 1**
Streamlines for steady, incompressible, axisymmetric, turbulent flow through a sudden contraction at $L_{extend}/D_2$ = 0.75.

*Upstream tube*:

$$\text{Re}_1 = \frac{\rho V_1 D_1}{\mu} = \frac{\left(998.2\text{ kg/m}^3\right)\left(0.05\text{ m/s}\right)\left(0.008\text{ m}\right)}{0.001003\text{ kg/m}\cdot\text{s}} = \mathbf{398.1} \quad (1)$$

*Downstream tube*:

$$\text{Re}_2 = \frac{\rho V_2 D_2}{\mu} = \frac{\left(998.2\text{ kg/m}^3\right)\left(0.8\text{ m/s}\right)\left(0.002\text{ m}\right)}{0.001003\text{ kg/m}\cdot\text{s}} = \mathbf{1592} \quad (2)$$

Since both of these values are smaller than 2300, the laminar flow assumption for Problem 15-37 is reasonable. For the turbulent pipe flow of the present problem,

*Upstream pipe*:

$$\text{Re}_1 = \frac{\rho V_1 D_1}{\mu} = \frac{\left(998.2\text{ kg/m}^3\right)\left(1.0\text{ m/s}\right)\left(0.8\text{ m}\right)}{0.001003\text{ kg/m}\cdot\text{s}} = \mathbf{796{,}200} \quad (3)$$

*Downstream pipe*:

$$\text{Re}_2 = \frac{\rho V_2 D_2}{\mu} = \frac{\left(998.2\text{ kg/m}^3\right)\left(16.0\text{ m/s}\right)\left(0.2\text{ m}\right)}{0.001003\text{ kg/m}\cdot\text{s}} = \mathbf{3{,}185{,}000} \quad (4)$$

These Reynolds numbers are clearly high enough that the flow is indeed turbulent.

($b$) Our CFD solutions reveal reversed flow for the smallest two cases, i.e. for $L_{extend}/D_2$ = 0.25 and 0.5. For higher values of $L_{extend}/D_2$ there is no reverse flow. Comparing to the results of Problem 15-37, apparently the separation bubble for turbulent flow is somewhat longer (proportionally) than that for laminar flow. This is verified by comparing the streamlines of Fig. 1 to those of Fig. 1c of Problem 15-37.

($c$) Pressures $P_{in}$, $P_1$, and $\Delta P = P_{in} - P_1$ are tabulated as functions of $L_{extend}/D_2$ in the table. Note that we use units of kPa instead of Pa here for convenience. $\Delta P$ is independent of $L_{extend}$ (to three significant digits of precision) by $L_{extend}/D_2$ = 0. 5.

| $L_{extend}/D_2$ | $P_{in}$ (kPa, gage) | $P_1$ (kPa, gage) | $\Delta P$ (kPa) |
|---|---|---|---|
| 0.25 | 309.31 | 122.121 | 187.189 |
| 0.5 | 276.188 | 90.3473 | 185.841 |
| 0.75 | 251.176 | 65.3649 | 185.811 |
| 1 | 237.423 | 51.5843 | 185.839 |
| 1.25 | 228.954 | 43.0877 | 185.866 |
| 1.5 | 223.423 | 37.4649 | 185.958 |
| 2 | 216.361 | 30.2144 | 186.147 |

***Discussion*** Newer versions of FlowLab may give slightly different results. The lack of scatter in the turbulent data for $\Delta P$ beyond $L_{extend}/D_2$ = 0.5 is a rather pleasant surprise; we might have expected more scatter than the laminar flow solution of Problem 15-37, since we have two additional nonlinear transport equations to solve, with their associated interactions that complicate the solution.

**15-41**

**Solution** We are to compare the pressure drop through a sudden contraction in a round pipe with two different outlet conditions: pressure outlet and outflow.

***Assumptions*** **1** The flow is incompressible. **2** The flow is axisymmetric about the *x* axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is water with $\rho$ = 998.2 kg/m$^3$ and $\mu$ = 0.001003 kg/m·s.

***Analysis*** For the previous case (pressure outlet boundary condition), the average pressure at the inlet is $P_{in}$ = **251.18 kPa**, the average pressure at $x$ = 0 is $P_1$ = **65.36 kPa**, and the pressure difference is $\Delta P = P_{in} - P_1$ = **185.82 kPa**. For the present case (outflow boundary condition), the average pressure at the inlet is $P_{in}$ = **127.94** kPa, the average pressure at $x$ = 0 is $P_1$ = **–57.87 kPa**, and the pressure difference is $\Delta P = P_{in} - P_1$ = **185.81 kPa**. While the actual values of pressure differ throughout the contraction, the *pressure difference* agrees to nearly five digits of precision for the two cases. Thus we conclude that **the outlet boundary condition has very little effect on this flow field**.

For the case with the pressure outlet boundary condition, the static pressure at the outlet of the computational domain is forced to be constant (zero gage pressure in the calculations of the previous problem). In the present case however, the outflow boundary condition does *not* fix static pressure – rather, it forces flow variables to level off as they approach the outlet boundary.

***Discussion*** Newer versions of FlowLab may give slightly different results. With a velocity inlet and an outflow outlet, we do not fix the value of pressure at either boundary. Instead, the CFD code assigns $P$ = 0 gage pressure at some (arbitrary) location in the flow field (the default location is the origin). Even though the inlet and outlet pressures differ significantly between the two cases, $\Delta P$ is identical to almost five significant digits of precision. This result again verifies the statement we made in Chap. 9: For incompressible flow, it is not pressure itself which is important to the flow field, but rather pressure *differences*.

**15-42**

**Solution** We are to generate CFD solutions for several downstream duct extension lengths to see the impact of the downstream boundary.

***Assumptions*** **1** The flow is incompressible. **2** The flow is two-dimensional and symmetric about the $x$ axis. **3** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is water with $\rho$ = 998.2 kg/m$^3$ and $\mu$ = 0.001003 kg/m·s.

***Analysis*** ($a$) Our CFD solutions reveal reversed flow for the smallest three cases, i.e. for $L_{extend}/D_2$ = 0.25, 0.5, and 0.75. For higher values of $L_{extend}/D_2$, there is no reverse flow. Comparing to the results of Problem 15-40, the separation bubble for 2-D flow is somewhat longer than that for axisymmetric flow. This is verified by comparing the streamlines of Fig. 1 to those of Fig. 1 of Problem 15-40.

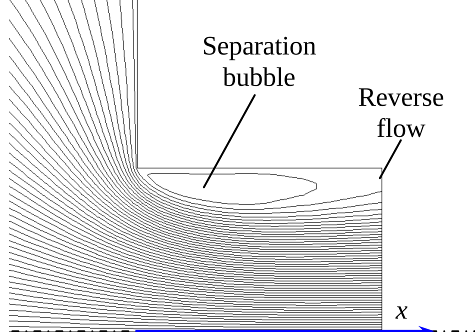



**FIGURE 1**
Streamlines for steady, incompressible, two-dimensional, turbulent flow through a sudden contraction at $L_{extend}/D_2$ = 0.75.

($b$) Pressures $P_{in}$, $P_1$, and $\Delta P = P_{in} - P_1$ are tabulated as functions of $L_{extend}/D_2$ in Table 1. $\Delta P$ is independent of $L_{extend}$ (to three significant digits of precision) by $L_{extend}/D_2$ = 0.5.

**TABLE 1**
Inlet pressure, average pressure at $x$ = 0, and pressure difference as functions of downstream extent of the computational domain. Data tabulated from CFD runs for steady, incompressible, two-dimensional, turbulent flow through a sudden contraction in a duct.

| $L_{extend}/D_2$ | $P_{in}$ (kPa, gage) | $P_1$ (kPa, gage) | $\Delta P$ (kPa) |
|---|---|---|---|
| 0.25 | 18.203 | 7.3241 | 10.8789 |
| 0.5 | 17.3614 | 6.65408 | 10.7073 |
| 0.75 | 15.3649 | 4.67913 | 10.6858 |
| 1 | 13.7764 | 3.09228 | 10.6841 |
| 1.25 | 12.9757 | 2.29043 | 10.6853 |
| 1.5 | 12.5034 | 1.81676 | 10.6867 |
| 2 | 11.9787 | 1.29006 | 10.6887 |
| 3 | 11.5309 | 0.845519 | 10.6853 |
| 4 | 11.3452 | 0.660043 | 10.6852 |

***Discussion*** Newer versions of FlowLab may give slightly different results. Comparing $\Delta P$ between this problem (2-D) and Problem 15-40 (axisymmetric), we see that $\Delta P$ for the axisymmetric case is more than 17 times greater than $\Delta P$ for the 2-D case. This is because for the axisymmetric case, the area downstream of the sudden contraction is 16 times smaller than the upstream area, while for the 2-D case, the area changes by only a factor of 4. Nevertheless, the 2-D case generates a longer separation bubble.

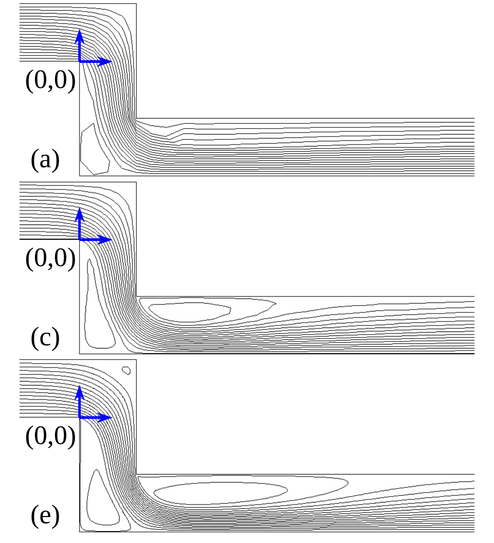
## 15-43

**Solution** We are to generate CFD solutions for several grid resolutions to test for grid independence for flow through a jog in a channel. Specifically, we are to compare streamlines and pressure difference at each level of grid resolution.

***Assumptions*** **1** The flow is incompressible and two-dimensional. **2** The flow is turbulent, but steady in the mean.

***Properties*** The fluid is air with $\rho = 1.225$ kg/m$^3$ and $\mu = 1.7894 \times 10^{-5}$ kg/m·s.

***Analysis***
(*a*) Streamlines are plotted for six grid resolution cases in Fig. 1. At the very coarse grid resolution, a small separation bubble appears at the lower left corner of the jog (Fig. 1a); flow separation is not obvious at the other corners, but it appears imminent at the most downstream corner. As grid resolution improves, the separation bubble on the lower left corner grows in size (compare Figs. 1a and 1b), and another separation bubble appears downstream of the jog. With continued improvement of the grid, both separation bubbles, especially the downstream one, continue to grow in size (Fig. 1c through 1f). Meanwhile, the streamlines become more rounded near the upper right corner of the jog. By 13,600 cells (Fig. 1e), a small recirculating zone is seen in this corner. The streamline pattern settles down by about 13,600 cells. We note that simulation of flow separation, separation bubbles, and reattachment is often a very difficult task for a CFD program. In this particular problem we must use a fairly fine grid in order to resolve all the details of the flow separation.



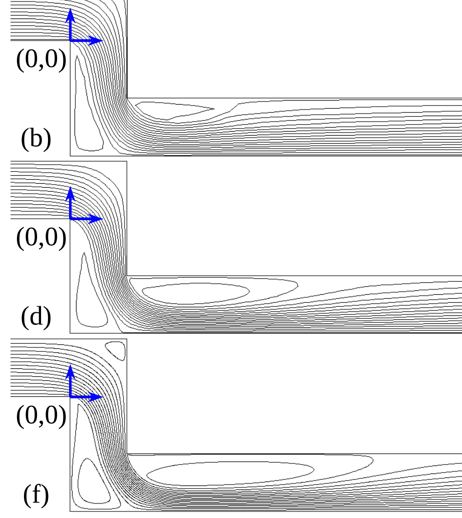




**FIGURE 1**
Streamlines for steady, incompressible, two-dimensional, turbulent flow through a jog in a channel at various levels of grid resolution: cell count = (a) 900, (b) 2500, (c) 5700, (d) 9100, (e) 13,600, and (f) 18,000. The origin (0,0) is marked on each figure for reference.

(*b*) $\Delta P$ is tabulated as a function of cell count in the table. As grid resolution improves, $\Delta P$ increases, reflecting the effect of the larger separation bubbles. Physically, we achieve less and less pressure recovery as the separation bubbles grow. We see that $\Delta P$ becomes independent of grid resolution to four significant digits by the fifth level of resolution, namely, at 13,600 cells. This is also seen in Fig. 2 where $\Delta P$ is plotted as a function of number of cells.

| cell count | $\Delta P$ (Pa) |
|---|---|
| 900 | 2.26846 |
| 2500 | 2.63636 |
| 5700 | 2.71292 |
| 9100 | 2.71739 |
| 13600 | 2.71835 |
| 15200 | 2.71828 |
| 18000 | 2.71856 |
| 20352 | 2.71746 |
| 24840 | 2.71544 |

**FIGURE 2**
Pressure difference vs. number of cells.

***Discussion*** Newer versions of FlowLab may give slightly different results. The sharp corners in the streamlines of the very coarse grid resolution case (Fig. 1a) are due to interpolation errors when the CFD code calculates contours of constant stream function. The outlet of the computational domain is reasonably far downstream (ten channel heights) to avoid reverse flow at the outlet.

**15-44**

**Solution** We are to generate CFD solutions for several grid resolutions to test for grid independence for flow through a jog in a channel. Specifically, we are to compare streamlines and pressure difference at each level of grid resolution.

***Assumptions*** **1** The flow is incompressible. **2** The flow is two-dimensional. **3** The flow is laminar and steady.

***Properties*** The fluid is water with $\rho$ = 998.2 kg/m$^3$ and $\mu$ = 0.001003 kg/m·s.

***Analysis***
(*a*) Streamlines are plotted for six grid resolution cases in Fig. 1.

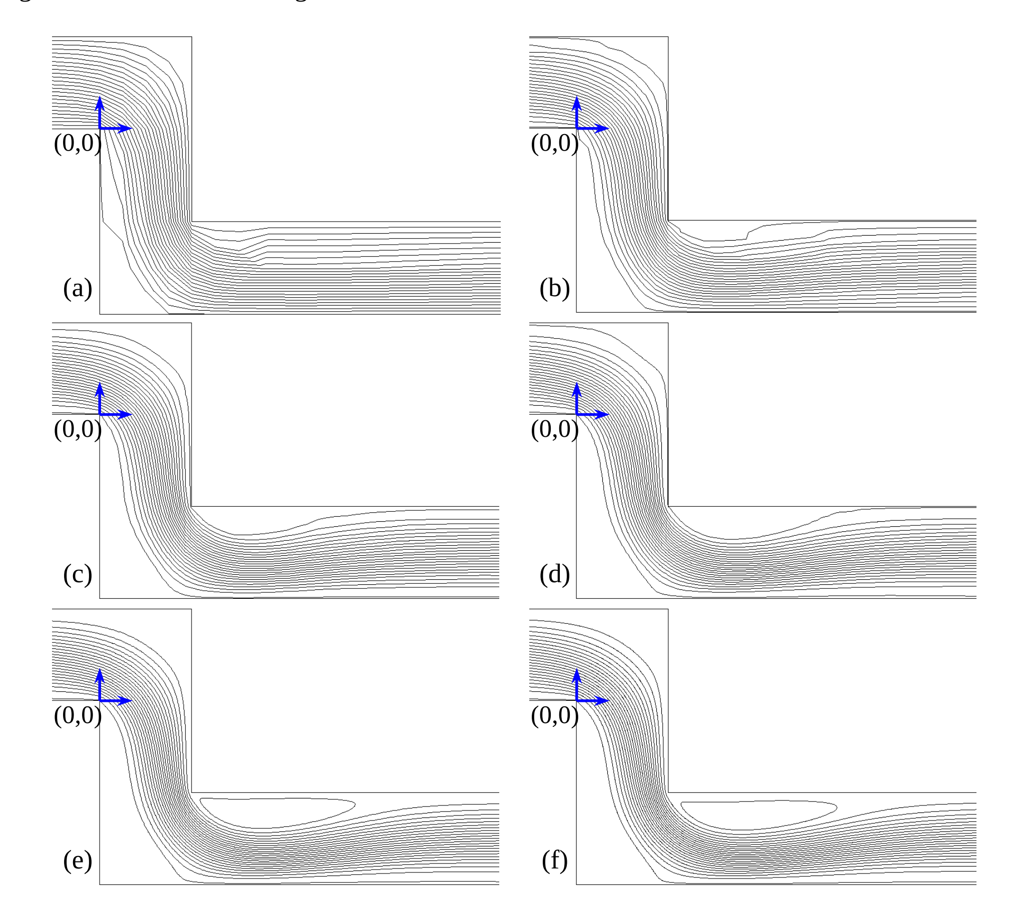



**FIGURE 1**
Streamlines for steady, incompressible, two-dimensional, laminar flow through a jog in a channel at various levels of grid resolution: cell count = (a) 272, (b) 576, (c) 847, (d) 1700, (e) 3388, and (f) 14,123. The origin (0,0) is marked on each figure for reference. The inlet velocity for these runs is 0.10 m/s.

At the very coarse grid resolution (Fig. 1a), the streamlines are not very smooth, but reveal the overall flow pattern. Although there are no closed streamlines, it appears that flow separation is imminent at the lower left and upper right corners of the jog. The streamlines appear fairly similar to those of the turbulent flow of the previous problem. At the grid resolution of Fig. 1b, the flow clearly separates at the sharp corner at the origin. The flow also separates at the downstream inside corner of the jog. The streamlines are not well-enough resolved to show closed separation bubbles. As the grid is further refines, the separation bubbles grow in size (compare Figs. 1b through 1d). In addition, the streamlines are smoother and more rounded. With continued improvement of the grid, the streamlines become more rounded near the upper right corner of the jog. By Fig. 1e, a recirculating flow pattern is seen downstream of the jog. **Based on streamline patterns, the grid is fully resolved at a cell count of about 10,000**. However, the pressure drop seems to be creeping up a bit as grid resolution is refined even further; it is not clear why.

(*b*) $\Delta P$ is tabulated as a function of grid resolution in Table 1. $\Delta P$ is also plotted as a function of number of cells in Fig. 2. As grid resolution improves, $\Delta P$ rises at first, reflecting the effect of the larger separation bubbles, but then decreases with further grid refinement, eventually leveling off after about 10,000 cells, implying **grid independence by 10$^4$ cells**.

***Discussion*** Newer versions of FlowLab may give slightly different results. All cases converge nicely – the flow at this Reynolds number is steady and laminar. There is no sign of instability in the flow. The outlet of the computational domain is reasonably far downstream (ten channel heights) to avoid reverse flow at the outlet.

**TABLE 1**
Pressure difference from inlet to outlet as a function of cell count, $V$ = 0.10 m/s.

| cell count | $\Delta P$ (Pa) |
|---|---|
| 272 | 37.6118 |
| 576 | 37.8096 |
| 847 | 37.5031 |
| 1700 | 36.9068 |
| 3388 | 36.5925 |
| 10625 | 36.4095 |
| 14123 | 36.4187 |
| 18645 | 36.435 |
| 23273 | 36.4636 |

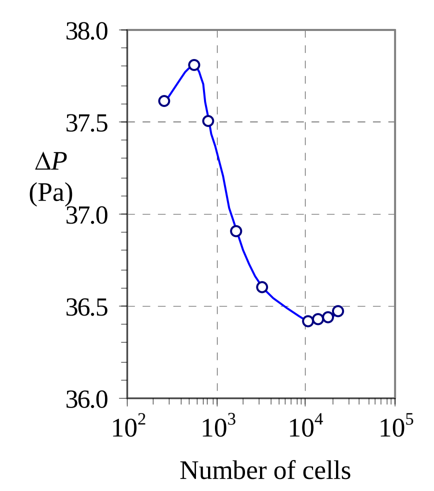



**FIGURE 2**
Pressure difference vs. number of cells, $V$ = 0.10 m/s.

**15-45**

**Solution** We are to generate CFD solutions for several grid resolutions to test for grid independence for flow through a jog in a channel. Specifically, we are to compare streamlines and pressure difference at each level of grid resolution.

***Assumptions*** **1** The flow is incompressible. **2** The flow is two-dimensional. **3** The flow is laminar and steady.

***Properties*** The fluid is water with $\rho$ = 998.2 kg/m$^3$ and $\mu$ = 0.001003 kg/m·s.

***Analysis***
(*a*) Streamlines are plotted for six grid resolution cases in Fig. 1.

**TABLE 1**
Pressure difference from inlet to outlet as a function of cell count for laminar flow through a two-dimensional jog, $V$ = 1.0 m/s.

| cell count | $\Delta P$ (Pa) |
|---|---|
| 272 | 2025.69 |
| 576 | 2269.26 |
| 847 | 2324.99 |
| 1700 | 2482.44 |
| 3388 | 2582.17 |
| 10625 | 2515.71 |
| 14123 | 2484.86 |
| 18645 | 2468.57 |
| 23273 | 2511.98 |

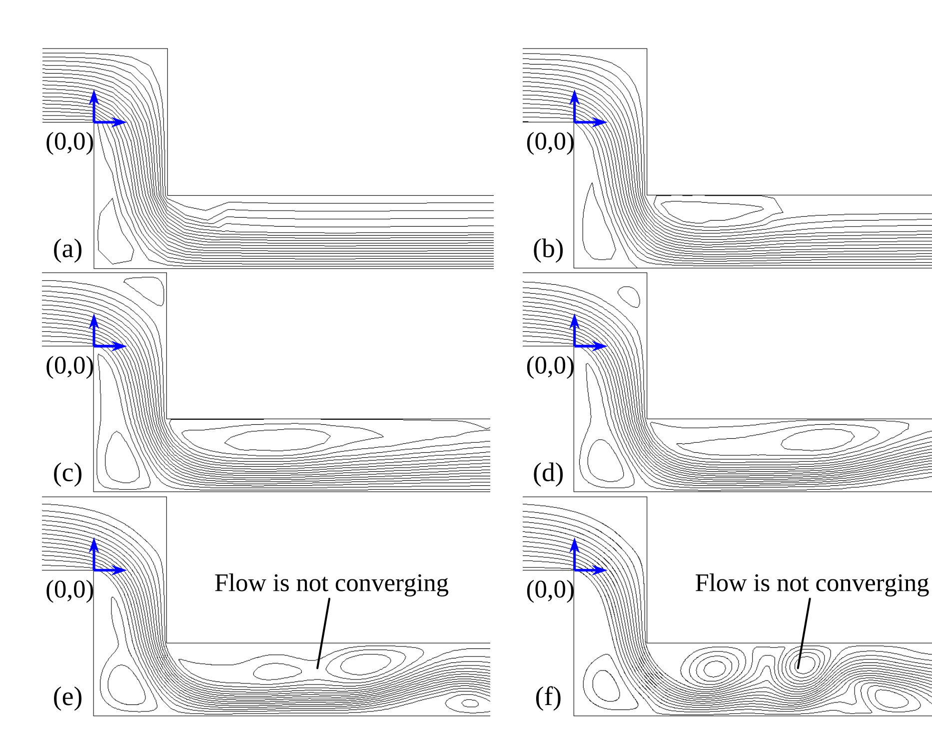



**FIGURE 1**
Streamlines at various levels of grid resolution: cell count = (a) 272, (b) 576, (c) 847, (d) 1700, (e) 3388, and (f) 14,123. The origin (0,0) is marked on each figure for reference. The inlet velocity for these runs is 1.0 m/s. For cases (e) and (f), the streamline patterns are at an arbitrary point in the solution. For these two cases, the CFD calculations do *not* converge, and eddies appear in the channel downstream of the jog. As the iterations continue, these eddies are swept downstream, but new ones appear, implying that the flow at this Reynolds number is not steady. The streamline patterns change as the CFD code iterates.

At the very coarse grid resolution, a small separation bubble appears at the lower left corner of the jog (Fig. 1a); flow separation is not obvious at the other corners, but it appears imminent at the most downstream corner. The streamlines appear quite similar to those of the turbulent flow of Problem 15-43. As grid resolution improves, the separation bubble on the lower left corner grows in size (compare Figs. 1a and 1b), the streamlines are smoother and more rounded, and another separation bubble appears downstream of the jog. With continued improvement of the grid, both separation bubbles continue to grow in size (Figs. 1c and 1d). Meanwhile, the streamlines become more rounded near the upper right corner of the jog. By Fig. 1c, a small recirculating zone is seen in this corner. The streamline pattern continues to change somewhat as the grid is further refined. However, just as we appear to be approaching grid independence, the flow develops some instabilities and **cannot converge to a steady-state solution**. This is first seen in the lower right portion of Fig. 1d – the streamlines drift away from the lower wall. While the streamline pattern in the front portion of the jog do not change appreciably, eddies develop in the recovery zone beyond the jog and are swept downstream as the grid is further refined (Figs. 1e and 1f). At the two finest grid resolutions, the flow is attempting to become unsteady. But since we are iterating towards a *steady* solution, the development and motion of these eddies is not physical. The solution fluctuates and is unable to converge. The bottom line is that we really should simulate this flow with an *unsteady* CFD solver. **Grid independence is not achieved because the flow becomes unsteady and unstable when a very high resolution grid is used**.

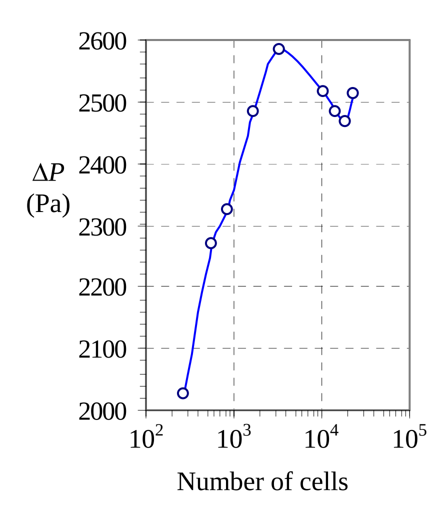



**FIGURE 2**
Pressure difference as a function of the number of cells in the computational domain; laminar flow through a jog in a channel. $V$ = 1.0 m/s.

(*b*) $\Delta P$ is tabulated as a function of grid resolution in Table 1. $\Delta P$ is also plotted as a function of number of cells in Fig. 2. As grid resolution improves, $\Delta P$ does not level off because the flow field is unstable – we should use an unsteady solver instead of a steady solver for this flow.

The Reynolds number for the present problem is

*Re at V = 1.0 m/s*:

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{\left(998.2 \text{ kg/m}^3\right)\left(1.0 \text{ m/s}\right)\left(0.001 \text{ m}\right)}{0.001003 \text{ kg/m}\cdot\text{s}} = \mathbf{995} \qquad (1)$$

and for the previous problem, it is

*Re at V = 0.10 m/s*:

$$\text{Re} = \frac{\rho VD}{\mu} = \frac{\left(998.2 \text{ kg/m}^3\right)\left(0.10 \text{ m/s}\right)\left(0.001 \text{ m}\right)}{0.001003 \text{ kg/m}\cdot\text{s}} = \mathbf{99.5} \qquad (2)$$

Since the Reynolds number of the present case is ten times larger than that of the previous case, and since it is almost 1000, it is not surprising that unsteadiness and instabilities in the flow start to develop at high grid resolutions.

***Discussion*** Newer versions of FlowLab may give slightly different results. The results here are similar to those of CFD simulations of flow around a circular cylinder. Namely, the flow is naturally unsteady, and leads to convergence difficulties at high grid resolution when the simulation is forced to be steady.

**15-46**

**Solution** We are to generate CFD solutions for compressible flow of air through a converging-diverging nozzle, and compare mass flow rate at various values of back pressure.

***Assumptions*** **1** The flow is steady and compressible. **2** The flow is axisymmetric. **3** The flow is approximated as inviscid.

***Properties*** The fluid is air with $k = 1.4$.

***Analysis*** We tabulate and plot $\dot{m}$ as a function of $P_b/P_{0,\text{inlet}}$ in Table 1 and Fig. 1 respectively.

**TABLE 1**
Pressure difference from inlet to outlet as a function of cell count for laminar flow through a two-dimensional jog, $V = 1.0$ m/s.

| $P_b/P_{0,\text{inlet}}$ | $\dot{m}$ (kg/s) |
|---|---|
| 0.454545 | 9.08358 |
| 0.545455 | 9.08356 |
| 0.636364 | 9.08355 |
| 0.727273 | 9.08357 |
| 0.818182 | 9.08356 |
| 0.909091 | 9.08354 |
| 0.954545 | 9.03556 |
| 0.977273 | 6.91699 |
| 0.988636 | 4.97062 |
| 0.990909 | 4.45686 |
| 0.995455 | 3.16752 |

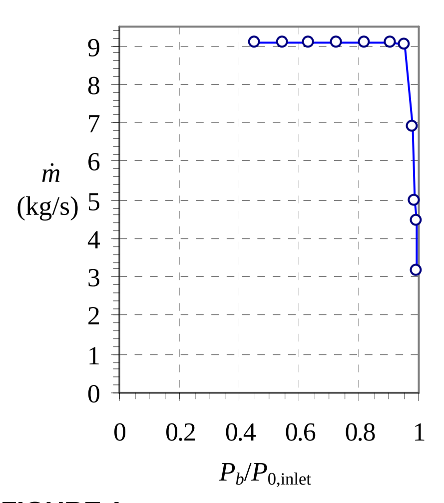



**FIGURE 1**
Mass flow rate as a function of back pressure ratio in a converging-diverging nozzle.

We see that $\dot{m}$ **is constant as long as the flow through the throat is sonic, because the flow is choked** ($P_b/P_{0,\text{inlet}} < 0.95$). For values of $P_b/P_{0,\text{inlet}}$ around 0.95 or higher, however, the flow in the diverging section becomes subsonic (no shock waves, and not choked), and the mass flow rate decreases with increasing $P_b/P_{0,\text{inlet}}$ from there on, as expected, since the flow is subsonic.

***Discussion*** The small variations in $\dot{m}$ in Table 1 (in the sixth digit of precision) are due to the fact that the residuals in the CFD solution do not go to zero. A finer grid and longer run times would correct this.

**15-47**

**Solution** We are to generate CFD solutions for compressible flow of air through a converging-diverging nozzle, and plot pressure and Mach number contours at two values of back pressure.

***Assumptions*** **1** The flow is steady and compressible. **2** The flow is axisymmetric. **3** The flow is approximated as inviscid.

***Properties*** The fluid is air with $k = 1.4$.

***Analysis*** We plot contours of $P$ and Ma in Fig. 1. For the case in which $P_b/P_{0,\text{inlet}} = 0.455$ (Fig. 1a), we observe a shock wave in the diverging portion of the nozzle. When $P_b/P_{0,\text{inlet}} = 0.977$ (Fig. 1b), however, the flow in the entire nozzle is subsonic, and there are no shock waves.

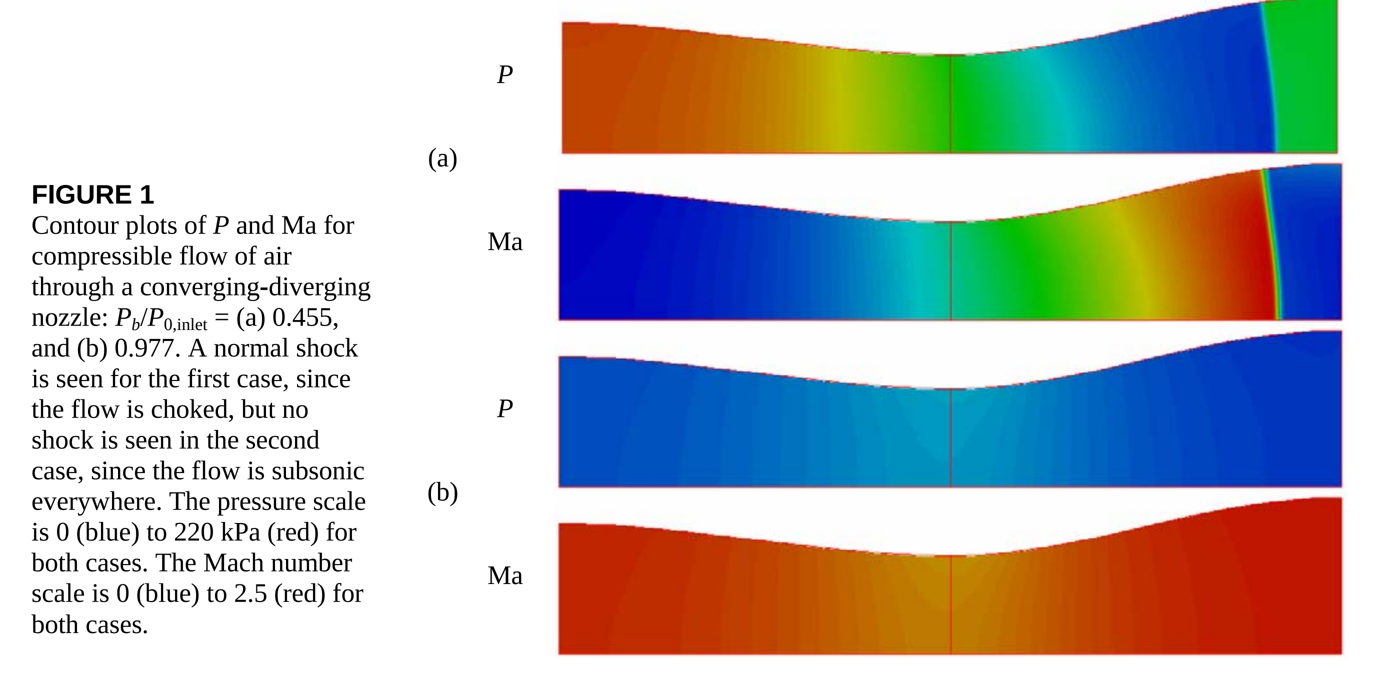



**FIGURE 1**
Contour plots of $P$ and Ma for compressible flow of air through a converging-diverging nozzle: $P_b/P_{0,\text{inlet}}$ = (a) 0.455, and (b) 0.977. A normal shock is seen for the first case, since the flow is choked, but no shock is seen in the second case, since the flow is subsonic everywhere. The pressure scale is 0 (blue) to 220 kPa (red) for both cases. The Mach number scale is 0 (blue) to 2.5 (red) for both cases.

Plots of $P$ and Ma versus $x$ are shown in Fig. 2 for the two cases. Also shown are calculations from one-dimensional inviscid theory. The agreement is excellent.

***Discussion*** Newer versions of FlowLab may give slightly different results. The shock wave calculated by CFD is not straight, but curved, since the calculations are axisymmetric, and therefore show more detail than the simplified one-dimensional approximation.

**15-48**

**Solution** We are to repeat the previous problem, but for the axisymmetric case.

***Analysis*** The results are expected to be similar, except that for the 2-D case, the area change is much less significant. Therefore it the value of back pressure that causes the flow to choke will be different than the axisymmetric case. We should still be able to observe a normal shock in the flow at low-enough values of back pressure.

***Discussion*** Newer versions of FlowLab may give slightly different results.

**15-49**

**Solution** We are to study the effect of rear-end shape on automobile drag coefficient.

***Analysis*** The template provides five different geometries. Model 1 has a very blunt rear end, kind of like a station wagon. As the model number increases, the rear end gets more slanted and rounded. Table 1 shows the calculated drag coefficient for each model.

It turns out that **Model number 4 has the lowest drag coefficient**, and Model number 1 has the highest. We probably would have guessed the latter, but not the former. Namely, the car with the most blunt rear end (Model 1) has the highest drag as we might expect, but most people would predict that Model 5, which is the most rounded, would have the lowest drag. Model 4 has a short notch for the trunk, and the aerodynamics turn out such that it has the lowest drag.

**TABLE 1**
Drag coefficient as a function of model number, where the rear end of the car is modified according to model number.

| Model number | Description | $C_D$ |
|---|---|---|
| 1 | Blunt rear end, like a station wagon | 0.320 |
| 2 | Back window with a short trunk section | 0.298 |
| 3 | Similar to model 2, but with a longer trunk | 0.276 |
| 4 | More rounded back end with a short notch for the trunk | 0.180 |
| 5 | Fully rounded back end | 0.212 |

Streamline plots are shown in Fig. 1 for Models 1 and 4, the highest and lowest drag cases, respectively. It is clear from these plots that the blunted rear body has a larger separation bubble in the wake (low pressure in the wake which leads to large drag). On the other hand, Model 4 has a much smaller wake and therefore much less drag.

| | Model | |
|---|---|---|
| (a) | 1 | |
| (b) | 4 | |

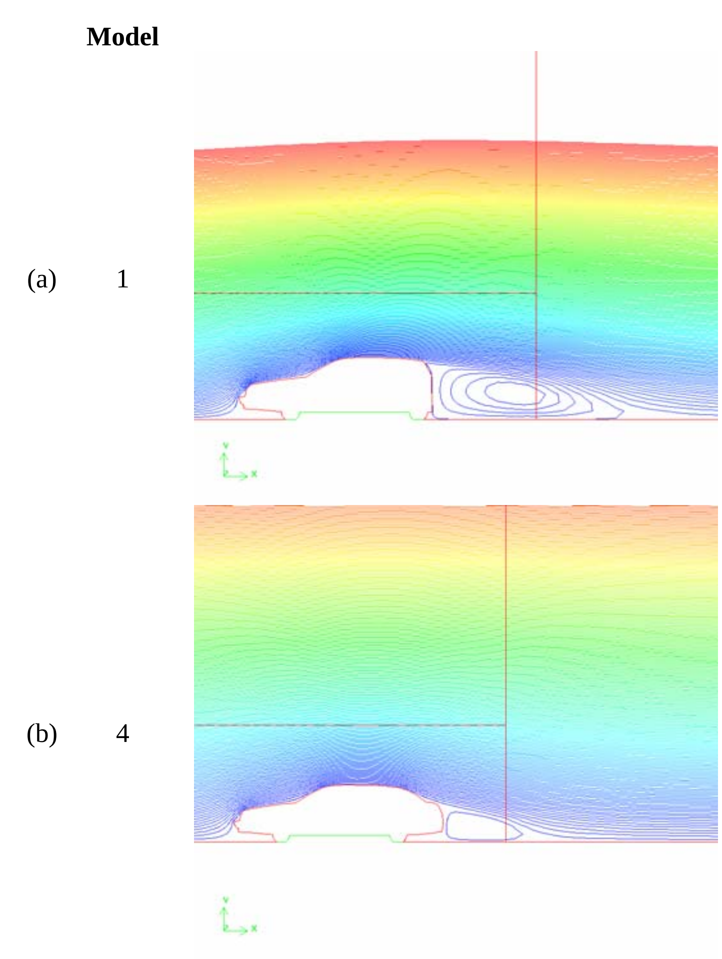


**FIGURE 1**
Streamlines for two representative two-dimensional automobile shapes: (a) Model 1, and Model 4.

***Discussion*** Newer versions of FlowLab may give slightly different results. It is not always immediately obvious whether a shape will have more or less drag than another shape. These results are two-dimensional, whereas actual automobiles, of course, are three-dimensional [see Problem 15-52].

## 15-50

**Solution** We are to examine the effect of the location of the upstream boundary on automobile drag calculations.

***Analysis*** We run five cases, and show the results in Table 1. It turns out that by about 20 heights away, the upper boundary condition no longer impacts the solution significantly to within three significant digits. The final drag coefficient is reported as 0.180 to three significant digits.

These data are also plotted in Fig. 1. We use a log scale on the horizontal axis since the range of $H/h$ is fairly large. For three digits of precision, it is necessary that the top boundary be at least 20 car heights tall. If $H/h$ is shorter than this, the upper boundary of the computational domain adversely influences the flow field. In the real-life flow, of course, the upper boundary is nearly infinite.

**TABLE 1**
Drag coefficient as a function of the normalized extent of the computational domain for turbulent flow over a cylindrical block.

| ***H/h*** | $C_D$ |
|---|---|
| 5 | 0.2101 |
| 10 | 0.1847 |
| 20 | 0.1798 |
| 35 | 0.1796 |
| 50 | 0.1797 |
| 100 | 0.1798 |

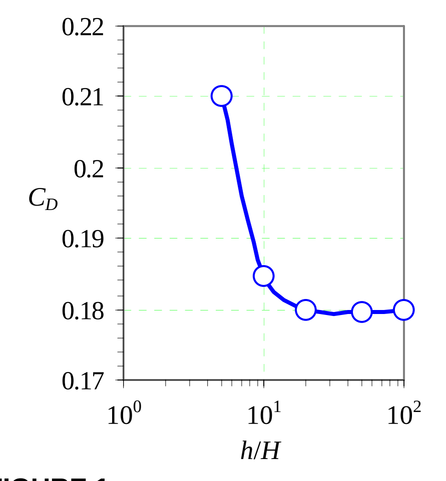



**FIGURE 1**
Drag coefficient plotted as a function of the normalized extent of the computational domain for turbulent flow over a two-dimensional automobile.

***Discussion*** Newer versions of FlowLab may give slightly different results. The bottom wall of the computational domain is not moving in these calculations, so the flow near the ground is not modeled properly. Furthermore, the calculations are two-dimensional, while a real car is of course fully three-dimensional.

## 15-51

**Solution** We are to compare turbulence models for the calculation of automobile drag.

***Analysis*** The results from the CFD calculations are presented in Table 1. There is some variation in the calculated values of $C_D$ depending on the turbulence model used; **$C_D$ ranges from 0.175 for the Spallart-Allmaras model to 0.223 for the RSM model**. The range of scatter is about 12%, which is actually not that large for comparison of four very different turbulence models. **It is impossible to say which one, if any, is correct**, since all turbulence models are approximations, with calibrated constants. Furthermore, we have no experimental data with which to compare the CFD results.

**TABLE 1**
Predicted drag coefficient on a two-dimensional automobile as a function of turbulence model.

| **Turbulence model** | $C_D$ |
|---|---|
| Spallart-Allmaras (1 eq.) | 0.175 |
| $k$-$\varepsilon$ (2 eq.) | 0.182 |
| $k$-$\omega$ (2 eq.) | 0.221 |
| Reynolds stress model (7 eq.) | 0.223 |

***Discussion*** Newer versions of FlowLab may give slightly different results. It would be good if we had experimental data with which to compare.

**15-52**

**Solution** We are to compare 2-D and 3-D drag predictions for flow over an automobile.

***Analysis*** The 3-D drag coefficient is **0.785**, significantly higher than the 2-D case, which is around 0.2 for most of the cases. This is most likely due to the fact that flow separates off the sides of the car as well as off the top. Note too that the car model used in this analysis is more like a truncated 2-D model rather than a truly three-dimensional model. If all the sharp corners were rounded off, the drag coefficient would be much lower than that calculated here.

***Discussion*** Newer versions of FlowLab may give slightly different results. This is not a very good drag coefficient for a modern car, and there is much improvement possible by further streamlining.

---

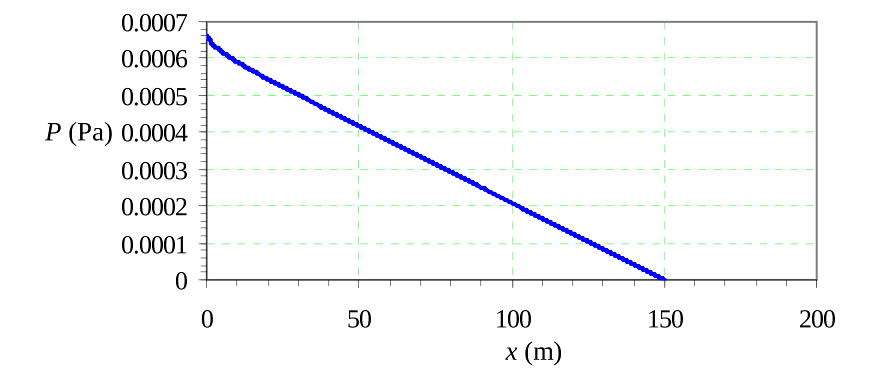
**15-53**

**Solution** We are to examine entrance the length for laminar pipe flow.

***Analysis*** Table 1 shows the entrance length calculations, along with a comparison with theory. The results are very good, as is expected for laminar pipe flow. Note that the results are somewhat subjective since the end of the developing region is being judged "by eye". Figure 1 shows the pressure along the axis as a function of downstream distance along the pipe axis for the case in which Re = 500. It is clear that the pressure drop is more severe (higher slope) at the beginning section of the pipe, but it is hard to tell at what axial location the flow becomes fully developed from this type of a plot. Thus, the entrance length is more appropriately determined by studying the velocity profiles, as above.

**TABLE 1**
Entry length vs. Reynolds number for developing laminar pipe flow.

| Re | *Le*/*D*, CFD | *Le*/*D*, Theory |
|---|---|---|
| 500 | 20 | 30 |
| 1000 | 50 | 60 |
| 1500 | 86 | 90 |
| 2000 | 1050 | 120 |



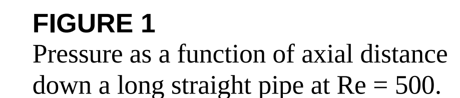



**FIGURE 1**
Pressure as a function of axial distance down a long straight pipe at Re = 500.

***Discussion*** Newer versions of FlowLab may give slightly different results. Also, student answers may vary considerably because of the subjectivity of these results, as mentioned above. There are no shed vortices, unsteadiness, or non-symmetries in straight laminar pipe flow, so it is no surprise that the CFD calculations perform so well.

---

## 15-54

**Solution** We are to study the entrance region in turbulent pipe flow.

***Analysis*** Table 1 shows the entrance length calculations, along with a comparison with the empirical equation. The CFD results are consistently low compared to the empirical results, although the agreement improves as Reynolds number increases. Since the grid is sufficiently resolved and $y^+$ values are appropriate, the discrepancies are due to deficiencies in the turbulence model.

***Discussion*** Newer versions of FlowLab may give slightly different results.

**TABLE 1**
Entry length vs. Reynolds number for developing turbulent pipe flow.

| Re | *Le*/*D*, CFD | *Le*/*D*, Exper. |
|---|---|---|
| 2000 | 10 | 15.6 |
| 5000 | 11 | 18.2 |
| 10000 | 13 | 20.4 |
| 20000 | 18 | 22.9 |
| 50000 | 22 | 26.7 |

## 15-55

**Solution** We are to study fully developed laminar pipe flow, and compare CFD calculations of Darcy friction factor with theory.

***Analysis*** Table 1 shows the CFD calculations of *f*, along with a comparison with theory. The CFD and theoretical results agree *very* well, since that the grid is well-enough resolved, and the flow is laminar. The values agree to within 0.05%, which is as good as one can ever expect from a CFD analysis.

**TABLE 1**
Darcy friction factor vs. Reynolds number for fully developed laminar pipe flow – comparison between CFD calculations and theory ($f$ = 64/Re).

| Re | *f*, CFD | *f*, Theory | Error (%) |
|---|---|---|---|
| 500 | 0.127944 | 0.128 | -0.04375 |
| 750 | 0.0852963 | 0.085333 | -0.0434 |
| 1000 | 0.0639721 | 0.064 | -0.04359 |
| 1250 | 0.0511774 | 0.0512 | -0.04414 |
| 1500 | 0.0426476 | 0.042667 | -0.04469 |
| 2000 | 0.0319855 | 0.032 | -0.04531 |

***Discussion*** Newer versions of FlowLab may give slightly different results. There are no shed vortices, unsteadiness, or non-symmetries in straight laminar pipe flow, so it is no surprise that the CFD calculations perform so well.

## 15-56

**Solution** We are to study fully developed turbulent pipe flow, with smooth walls and compare CFD calculations of Darcy friction factor with experimentally determined values.

***Analysis*** Table 1 shows the CFD calculations of *f*, along with a comparison with the empirical formula (the Colebrook equation). The CFD and theoretical results agree reasonably well (within less than 10% for all cases tested), and the agreement improves with increasing Reynolds number. The $k$-$\varepsilon$ model performs better at the higher values of Reynolds number because the turbulence model is calibrated for high Re flows.

**TABLE 1**
Darcy friction factor vs. Reynolds number for fully developed turbulent pipe flow.

| Re | *f*, CFD | *f*, Exper. |
|---|---|---|
| 5000 | 0.040007 | 0.0374 |
| 10000 | 0.03293 | 0.0309 |
| 50000 | 0.022029 | 0.0209 |

***Discussion*** Newer versions of FlowLab may give slightly different results. Ten percent agreement is excellent, considering that the Colebrook equation (or the Moody chart) is accurate to only about 15% to begin with.

## 15-57

**Solution** We are to study fully developed turbulent pipe flow with rough walls, and compare CFD calculations of Darcy friction factor with experimentally determined values.

***Analysis*** Table 1 shows the CFD calculations of *f*, along with a comparison with the empirical formula (the Colebrook equation). The CFD and theoretical results agree reasonably well (within less than 15% for all cases tested), and the agreement improves with increasing roughness. Since the agreement is within the known inaccuracy of the Colebrook equation (about 15%), these CFD results are considered adequate.

***Discussion*** Newer versions of FlowLab may give slightly different results.

**TABLE 1**
Darcy friction factor vs. dimensionless roughness height for fully developed turbulent pipe flow at Re = $1 \times 10^6$.

| $\varepsilon/D$ | *f*, CFD | *f*, Exper. |
|---|---|---|
| 0.00005 | 0.0113382 | 0.0126 |
| 0.00025 | 0.0134436 | 0.0152 |
| 0.0005 | 0.016288 | 0.0172 |
| 0.001 | 0.0204516 | 0.0199 |
| 0.0015 | 0.0231213 | 0.0220 |
| 0.002 | 0.0246173 | 0.0236 |

## 15-58

**Solution** We are to model the laminar boundary layer on a flat plate using CFD, and compare to analytical results.

***Analysis*** For a Reynolds number of $1 \times 10^5$, the CFD calculations give a nondimensional boundary layer thickness of $\delta/x$ = **0.0154** and a drag coefficient of $C_D$ = **0.00435**. The theoretical values are obtained from equations in Chaps. 10 and 11, namely,

$$\frac{\delta}{x} = \frac{4.91}{\sqrt{\text{Re}_x}} = \frac{4.91}{\sqrt{1\times 10^5}} = 0.0155 \qquad (1)$$

and

$$C_D = C_f = \frac{1.33}{\sqrt{\text{Re}_x}} = \frac{1.33}{\sqrt{1\times 10^5}} = 0.00421 \qquad (2)$$

The agreement is excellent for both values, the discrepancy being less than 1% for $\delta/x$ and about 3% for drag coefficient.

***Discussion*** Newer versions of FlowLab may give slightly different results.

## 15-59

**Solution** We are to compare CFD results to experimental results for the case of a flat plate turbulent boundary layer.

***Analysis*** For a Reynolds number of $1 \times 10^7$, the CFD calculations give a nondimensional boundary layer thickness of $\delta/x$ = **0.0140** and a drag coefficient of $C_D$ = **0.00292**. The empirical values are obtained from equations in Chaps. 10 and 11, namely,

$$\frac{\delta}{x} = \frac{0.38}{\text{Re}_x^{1/5}} = \frac{0.38}{\left(1\times 10^7\right)^{1/5}} = 0.015 \qquad \text{and} \qquad C_D = C_f = \frac{0.074}{\text{Re}_x^{1/5}} = \frac{0.074}{\left(1\times 10^7\right)^{1/5}} = 0.0029$$

The agreement is excellent for both values, the discrepancy being about 7% for $\delta/x$ and negligible (within 2 significant digits) for drag coefficient.

***Discussion*** Newer versions of FlowLab may give slightly different results. We report our empirical values to only two significant digits in keeping with the level of precision and accuracy of turbulent flows.

## 15-60

**Solution** We are to compare turbulence models in CFD calculations of a flat plate turbulent boundary layer.

***Analysis*** The calculations for drag coefficient and boundary layer thickness are shown in Table 1 for four turbulence models: standard $k$-$\varepsilon$, standard $k$-$\omega$, Spallart-Allmaras, and the Reynolds stress model (RSM). The first two are two-equation models, the third is a one-equation model, and the fourth is a full Reynolds stress model (5 equations for the two-dimensional case). All turbulence models are approximations, with calibrated constants. The drag coefficients range from 0.0027 for the RSM model to 0.0029 for the $k$-$\varepsilon$ model. The empirical value is 0.0029. The dimensionless boundary layer thickness ranges from 0.012 for the $k$-$\omega$ model to 0.014 for the $k$-$\varepsilon$ model. The empirical value is 0.015. Thus, all the turbulence models do very well at predicting this flow, and the $k$-$\varepsilon$ model performs the best, overall. The RSM model, in spite of its increased complexity, does not do as well as some of the simpler models. Since all turbulence models are calibrated with the turbulent flat plate boundary layer, it is not surprising that all of them give reasonable results.

**TABLE 1**
Drag coefficient and boundary layer thickness for a flat plate at $Re = 1 \times 10^7$, as predicted by CFD with various turbulence models.

| $\varepsilon/D$ | $C_D$ | $\delta/L$ |
|---|---|---|
| $k$-$\varepsilon$ | 0.00291589 | 0.0140152 |
| $k$-$\omega$ | 0.00281271 | 0.0122085 |
| S-A | 0.00276828 | 0.013329 |
| RSM | 0.00274706 | 0.0131455 |

***Discussion*** Newer versions of FlowLab may give slightly different results.

## 15-61

**Solution** We are to compare the velocity and thermal boundary layer thicknesses on a heated flat plate, laminar flow.

***Analysis*** The results are shown in Table 1. We denote the thermal boundary layer thickness as $\delta_T/L$, and compare to the velocity boundary layer thickness, which we denote as $\delta_u/L$. The velocity (or momentum) boundary layer thicknesses are identical, as expected, since the Reynolds number is the same for either fluid. However, since the Prandtl number ($Pr = \nu/\kappa$) of water is close to 1000, which is much greater than one, the velocity boundary layer thickness is much greater than the thermal boundary layer thickness for the water flow. In other words, momentum diffuses into the free-stream flow much more rapidly than does temperature since the Prandtl number is large. The two thicknesses are nearly equal for the air, although the thermal boundary layer thickness is somewhat higher than the momentum boundary layer thickness. This is to be expected since the Prandtl number in air is about 0.70, which means that temperature diffuses more rapidly than does momentum since Pr < 1.

**TABLE 1**
Nondimensional temperature and boundary layer thicknesses for a heated flat plate at $Re = 1 \times 10^5$, as predicted by CFD (laminar flow).

| Fluid | $\delta_T/L$ | $\delta_u/L$ |
|---|---|---|
| air | 0.017313 | 0.015412 |
| water | 0.007664 | 0.015412 |

***Discussion*** Newer versions of FlowLab may give slightly different results. Since the flow is laminar and the grid is well resolved with a large computational domain, these results are nearly "exact".

## 15-62

**Solution** We are to compare the velocity and thermal boundary layer thicknesses on a heated flat plate, turbulent flow.

***Analysis*** The results are shown in Table 1. We denote the thermal boundary layer thickness as $\delta_T/L$, and compare to the velocity boundary layer thickness, which we denote as $\delta_u/L$. The velocity (or momentum) boundary layer thicknesses are identical, as expected, since the Reynolds number is the same for either fluid. The velocity boundary layer thickness is slightly greater than the thermal boundary layer thickness for the water flow, while the two thicknesses are nearly equal for the air, although the thermal boundary layer thickness is slightly greater than the momentum boundary layer thickness. These small differences are attributed to differences in the Prandtl number, as explained for the laminar case of the previous problem. However, turbulent diffusion dominates over laminar diffusion, and therefore the effect of Prandtl number is minimal. The thermal and momentum boundary layer thicknesses are nearly equal in both fluids since turbulent diffusion effects dominate laminar (molecular) diffusion effects, especially at high Re.

**TABLE 1**
Nondimensional temperature and boundary layer thicknesses for a heated flat plate at Re = $1 \times 10^7$ , as predicted by CFD (turbulent flow, using the standard $k$-$\varepsilon$ turbulence model).

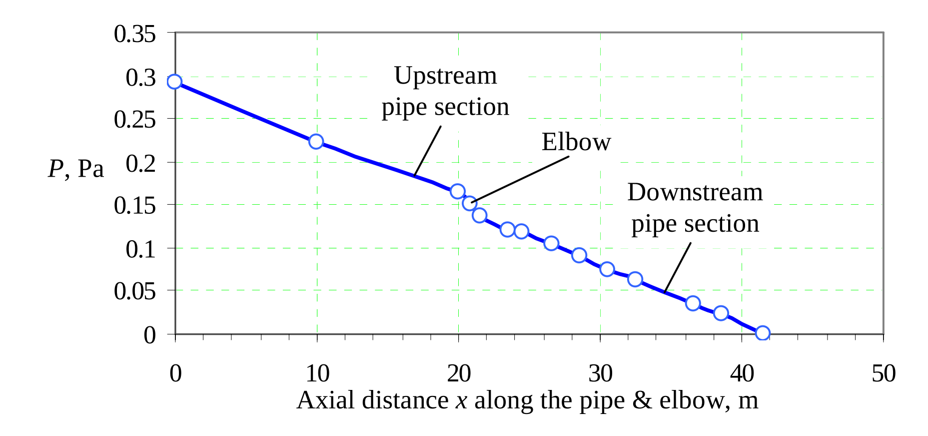
| Fluid | $\delta_T/L$ | $\delta_u/L$ |
|---|---|---|
| air | 0.014527 | 0.014015 |
| water | 0.013162 | 0.014014 |

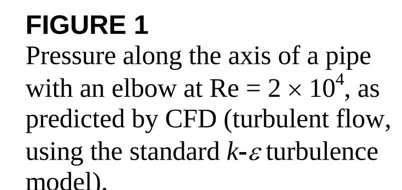
***Discussion*** Newer versions of FlowLab may give slightly different results. Mass (species), momentum (velocity), and energy (temperature) diffuse nearly equally in a turbulent flow since the large turbulent eddies cause rapid mixing through the boundary layer. Turbulence models do not actually calculate the details of the unsteady flow caused by these turbulent eddies; rather, they model the increased diffusion effects with approximations that enable the calculations to be performed in reasonable time on a computer.

## 15-63

**Solution** We are to calculate and plot pressure drop down a pipe with an elbow in turbulent flow.

***Analysis*** A plot of pressure as a function of axial distance along the upstream pipe, through the elbow, and through the downstream pipe is shown in Fig. 1. The pressure losses are nearly linear through the entrance region. In the elbow itself, and just downstream of it, the pressure drops rapidly. The rate of pressure drop in the pipe section *downstream* of the elbow is about the same as that upstream. Therefore, it appears that most of the pressure drop occurs **near the region of the elbow**.





**FIGURE 1**
Pressure along the axis of a pipe with an elbow at Re = $2 \times 10^4$, as predicted by CFD (turbulent flow, using the standard $k$-$\varepsilon$ turbulence model).

***Discussion*** Newer versions of FlowLab may give slightly different results.

## 15-64

**Solution** We are to study the counter-rotating vortices in a turbulent pipe flow downstream of an elbow.

***Analysis*** Velocity vectors at several cross sections are plotted in Fig. 1. **There are no counter-rotating eddies upstream of the elbow**. However, **they are formed as the fluid passes through the elbow, and are very strong just downstream of the elbow**. These vortices **decay in strength down the pipe after the elbow**, but they persist for a very long time, and may influence the accuracy of flow meters downstream of an elbow. This is why many manufacturers of pipe flow meters recommend that their flow meter be installed at least 10 or 20 pipe diameters downstream of an elbow – **to avoid influence of the counter-rotating eddies**.

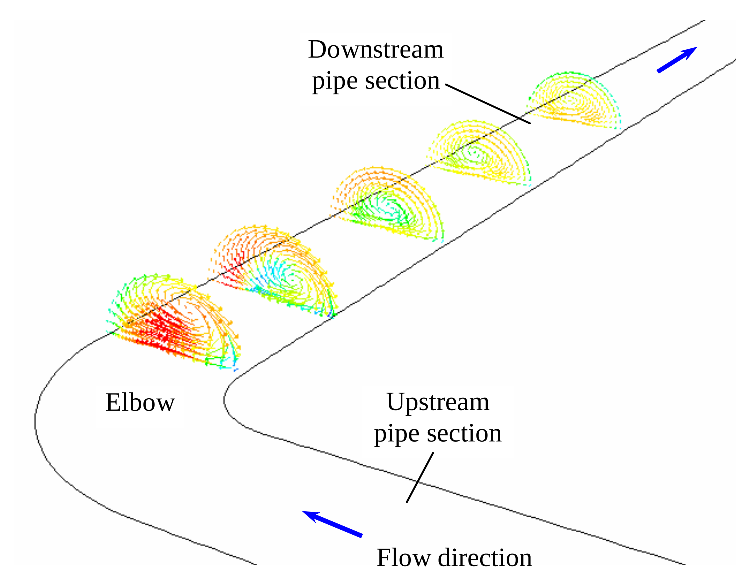



**FIGURE 1**
Velocity vector plots at several cross sections of a pipe with an elbow at Re = $2 \times 10^4$, as predicted by CFD (turbulent flow, using the standard $k$-$\varepsilon$ turbulence model).

***Discussion*** The counter-rotating eddies lead to additional irreversible head loss as they dissipate.

---

## 15-65

**Solution** We are to calculate the minor loss coefficient through a pipe elbow in turbulent flow.

***Analysis*** The value of $K_L$ given in Chap. 8 is **0.30**. For the pipe with the elbow, the pressure drop calculated by the CFD code is 0.284 kPa. For the straight pipe, the pressure drop is calculated to be 0.224 kPa. Subtracting these and converting to minor loss coefficient, the value of $K_L$ predicted by our CFD calculation for the standard $k$-$\varepsilon$ turbulence model is **0.295** – a difference of less than two percent, and well within the accuracy of the Colebrook equation, which is about 15%, and the accuracy of the tabulated minor loss coefficients, which is often much greater than 15%. This agreement is better than expected, considering that this is a very complex 3-D flow, and turbulence models may not necessarily apply for such problems.

***Discussion*** The agreement here is excellent – CFD does not always match so well with experiment, especially in turbulent flow when using turbulence models, since turbulence models are *approximations* that often lead to significant error.

---

## 15-66

**Solution** We are to compare various turbulence models – how well they predict the minor loss in a pipe elbow.

***Analysis*** The results for four turbulence models are listed in Table 1. **The standard *k*-$\varepsilon$ turbulence model does the best job** in predicting $K_L$. **The standard *k*-$\omega$ turbulence model does the worst job**. The Spallart-Allmaras turbulence model calculations are not the worst, even though this is the simplest of the models. One would hope that the more complicated model (Reynolds stress model) would do a better job than the simpler models, but this is not the case in the present problem (it does worse than the *k*-$\varepsilon$ model, but better than the other two. All turbulence models are approximations, with calibrated constants. While one model may do a better job in a certain flow, it may not do such a good job in another flow. This is the unfortunate state of affairs concerning turbulence models.

**TABLE 1**
Minor loss coefficient as a function of turbulence model for flow through a 90° elbow in a pipe. The error is in comparison to the experimental value of 0.30.

| Turbulence model | $K_L$ | Error (%) |
|---|---|---|
| Spallart-Allmaras (1 eq.) | 0.204 | -32% |
| *k*-$\varepsilon$ (2 eq.) | 0.295 | -1.7% |
| *k*-$\omega$ (2 eq.) | 0.401 | 34% |
| Reynolds stress model (7 eq.) | 0.338 | 13% |

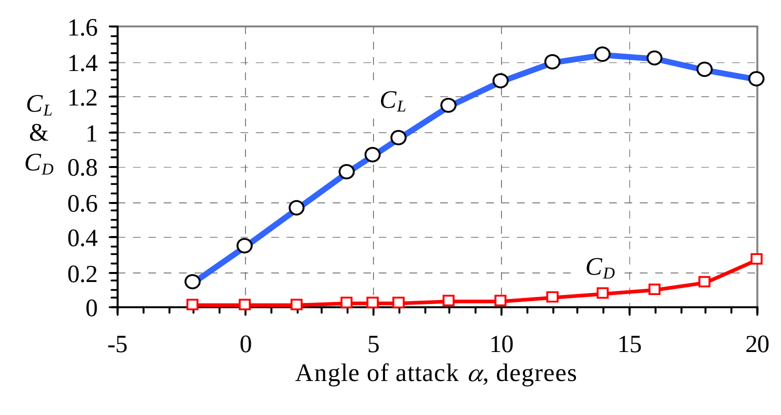
***Discussion*** Newer versions of FlowLab may give slightly different results. Although the RSM model does not seem to be too impressive based on this comparison, keep in mind that this is a very simple flow field. There are flows (generally flows of very complex geometries and rotating flows) for which the RSM model does a much better job than any 1- or 2-equation turbulence model, and is worth the required increase in computer resources.

## 15-67

**Solution** We are to use CFD to calculate the lift and drag coefficients on an airfoil as a function of angle of attack.

***Analysis*** The CFD analysis involves turbulent flow, using the standard *k*-$\varepsilon$ turbulence model. The results are tabulated and plotted. The lift coefficient rises to 1.44 at $\alpha$ = 14°, beyond which the lift coefficient drops off. So, **the stall angle is about 14°**. Meanwhile, the drag coefficient increases slowly up to the stall location, and then rises significantly after stall.

| $\alpha$ (degrees) | $C_L$ | $C_D$ |
|---|---|---|
| -2 | 0.138008 | 0.0153666 |
| 0 | 0.348498 | 0.0148594 |
| 2 | 0.560806 | 0.0149519 |
| 4 | 0.769169 | 0.0170382 |
| 5 | 0.867956 | 0.0192945 |
| 6 | 0.967494 | 0.0210042 |
| 8 | 1.14544 | 0.0275433 |
| 10 | 1.29188 | 0.0375832 |
| 12 | 1.39539 | 0.0522318 |
| 14 | 1.44135 | 0.0725146 |
| 16 | 1.41767 | 0.100056 |
| 18 | 1.34726 | 0.140424 |
| 20 | 1.29543 | 0.274792 |




***Discussion*** We note that this airfoil is not symmetric, as can be verified by the fact that the lift coefficient is nonzero at zero angle of attack. The lift coefficient does not drop as dramatically as is observed empirically. Why? The flow becomes *unsteady* for angles of attack beyond the stall angle. However, we are performing *steady* calculations. For higher angles, the run does not even converge; the CFD calculation is stopped because it has exceeded the maximum number of allowable iterations, not because it has converged. Thus the main reason for not capturing the sudden drop in $C_L$ after stall is because we are not accounting for the transient nature of the flow. The airfoil used in these calculations is called a *ClarkY airfoil*.

## 15-68

**Solution** We are to analyze the effect of Reynolds number on lift and drag coefficient.

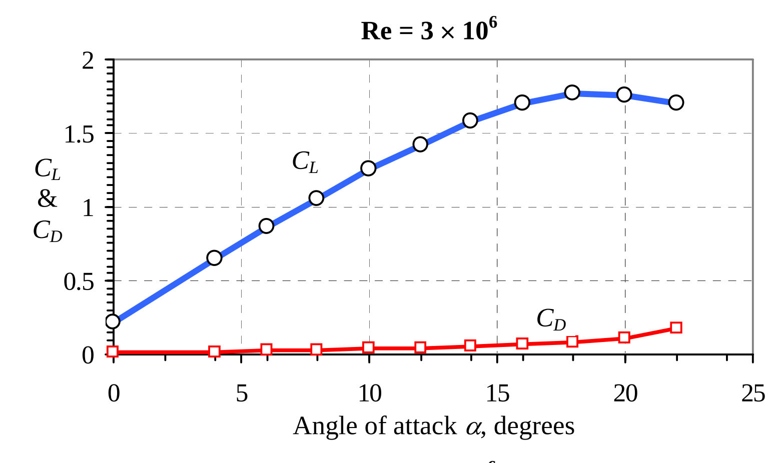
***Analysis*** The CFD analysis involves turbulent flow, using the standard $k$-$\varepsilon$ turbulence model. For this airfoil, which is different than the airfoil analyzed in the previous problem, and for the case in which Re = $3 \times 10^6$, the lift coefficient rises to about 1.77 at about 18°, beyond which the lift coefficient drops off (see the first table). So, **the stall angle is about 18°**. Meanwhile, the drag coefficient increases slowly up to the stall location, and then rises significantly after stall. The data are also plotted below.

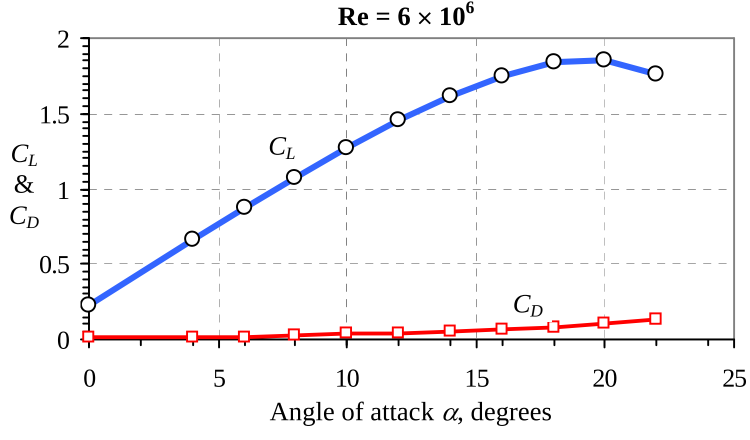
**Re = $3 \times 10^6$**

| $\alpha$ (degrees) | $C_L$ | $C_D$ |
|---|---|---|
| 0 | 0.221797 | 0.0118975 |
| 4 | 0.65061 | 0.0166523 |
| 6 | 0.858744 | 0.0212052 |
| 8 | 1.05953 | 0.0273125 |
| 10 | 1.2501 | 0.0351061 |
| 12 | 1.42542 | 0.0447038 |
| 14 | 1.57862 | 0.0562746 |
| 16 | 1.69816 | 0.0702321 |
| 18 | 1.76686 | 0.0875881 |
| 20 | 1.75446 | 0.111326 |
| 22 | 1.70497 | 0.178404 |

**Re = $6 \times 10^6$**

| $\alpha$ (degrees) | $C_L$ | $C_D$ |
|---|---|---|
| 0 | 0.226013 | 0.0106894 |
| 4 | 0.659469 | 0.015384 |
| 6 | 0.870578 | 0.0198087 |
| 8 | 1.07512 | 0.0256978 |
| 10 | 1.27094 | 0.0331355 |
| 12 | 1.4533 | 0.0422083 |
| 14 | 1.61589 | 0.0530114 |
| 16 | 1.74939 | 0.0657999 |
| 18 | 1.83901 | 0.0812925 |
| 20 | 1.85799 | 0.101563 |
| 22 | 1.76048 | 0.138806 |

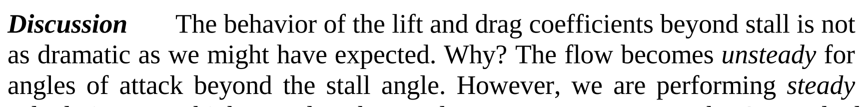



For the case in which Re = $6 \times 10^6$, lift coefficient rises to about 1.86 at about 20°, beyond which the lift coefficient drops off (see the second table). So, **the stall angle is about 20°**. Meanwhile, the drag coefficient increases slowly up to the stall location, and then rises significantly after stall. These data are also plotted.

The maximum lift coefficient and the stall angle have both increased somewhat compared to those at Re = $3 \times 10^6$ (half the Reynolds number). Apparently, the higher Reynolds number leads to a more vigorous turbulent boundary layer that is able to resist flow separation to a greater downstream distance than for the lower Reynolds number case. For all angles of attack, the drag coefficient is slightly smaller for the higher Reynolds number case, reflecting the fact that the skin friction coefficient decreases with increasing Re along a wall, all else being equal. [Airfoil drag (before stall) is due mostly to skin friction rather than pressure drag.]

Finally, we plot the lift coefficient as a function of angle of attack for the two Reynolds numbers. The airfoil clearly performs better at the higher Reynolds number.



***Discussion*** The behavior of the lift and drag coefficients beyond stall is not as dramatic as we might have expected. Why? The flow becomes *unsteady* for angles of attack beyond the stall angle. However, we are performing *steady* calculations. For higher angles, the run does not even converge; the CFD calculation is stopped because it has exceeded the

maximum number of allowable iterations, not because it has converged. Thus the main reason for not capturing the sudden drop in $C_L$ after stall is because we are not accounting for the transient nature of the flow. We note that the airfoil used in this problem (a *NACA2415 airfoil*) is different than the one used in Problem 15-67 (a *ClarkY airfoil*). Comparing the two, the present one has better performance (higher maximum lift coefficient and higher stall angle, even though the Reynolds numbers here are lower than that of Problem 15-67. At higher Re, this airfoil may perform even better.

---

## 15-69

**Solution** We are to examine the effect of grid resolution on airfoil stall at a given angle of attack and Reynolds number.

***Analysis*** The CFD results are shown in Table 1 for the case in which the airfoil is at a 15° angle of attack at a Reynolds number of $1 \times 10^7$. The lift coefficient levels off to a value of **1.44** to three significant digits by a cell count of about 17,000. The drag coefficient levels off to a value of **0.849** to three significant digits by a cell count of about 22,000. Thus, we have shown how far the grid must be refined in order to achieve grid independence. Thus, **we have achieved grid independence for a cell count greater than about 20,000**.

As for the effect of grid resolution on stall angle, we see that with poor grid resolution, flow separation is not predicted accurately. Indeed, when the grid resolution is poor (under 10,000 cells in this particular case), stall is not observed even though the angle of attack (15°) is above the stall angle (14°) for this airfoil at this Reynolds number ($1 \times 10^7$). When the cell count is about 15,000, however, stall *is* observed. Thus, **yes, grid resolution *does* affect calculation of the stall angle – it is not predicted well unless the grid is sufficiently resolved**.

**TABLE 1**
Lift and drag coefficients as a function of cell count (higher cell count means finer grid resolution) for the case of flow over a 2-D airfoil at an angle of attack of 15° and Re = $1 \times 10^7$.

| Cells | $C_L$ | $C_D$ |
|---|---|---|
| 672 | 1.09916 | 0.242452 |
| 1344 | 1.00129 | 0.224211 |
| 2176 | 1.06013 | 0.196212 |
| 6264 | 1.02186 | 0.189023 |
| 12500 | 1.42487 | 0.0869799 |
| 16800 | 1.4356 | 0.0857791 |
| 21700 | 1.43862 | 0.0848678 |
| 24320 | 1.43719 | 0.0852304 |
| 27200 | 1.43741 | 0.0854769 |

***Discussion*** Newer versions of FlowLab may give slightly different results.

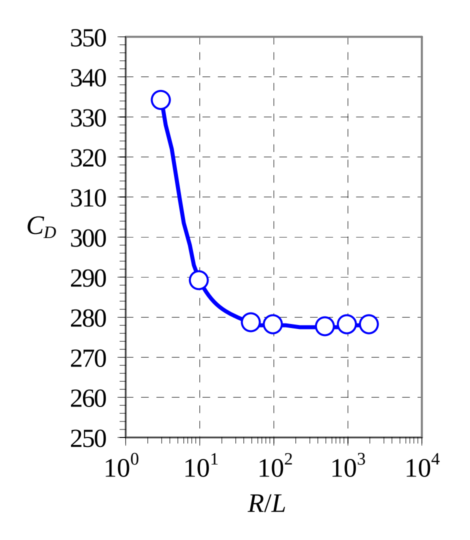
---

## 15-70

**Solution** We are to study the effect of computational domain extent on the calculation of drag in creeping flow.

***Analysis*** The drag coefficient is listed as a function of *R*/*L* in the table. The data are also plotted. From these data we see that for *R*/*L* greater than about 50, the drag coefficient has leveled off to a value of about **278** to three significant digits. This is rather surprising since the Reynolds number is so small, and the viscous effects are expected to influence the flow for tens of body lengths away from the body.

| *R*/*L* | $C_D$ |
|---|---|
| 3 | 334.067 |
| 10 | 289.152 |
| 50 | 278.291 |
| 100 | 277.754 |
| 500 | 277.647 |
| 1000 | 277.776 |
| 2000 | 278.226 |




***Discussion*** When analyzing creeping flow using CFD, it is important to extend the computational domain very far from the object of interest, since viscous effects influence the flow very far from the object. This effect is not as great at high Reynolds numbers, where the inertial terms dominate the viscous terms.

---

## 15-71

**Solution** We are to study the effect of Reynolds number on flow over an ellipsoid.

**TABLE 1**
Drag coefficient as a function of Reynolds number for flow over a 2×1 ellipsoid.

| Re | $C_D$ |
|---|---|
| 0.1 | 552.89 |
| 0.5 | 114.634 |
| 5 | 14.3342 |
| 20 | 4.89852 |
| 50 | 2.61481 |
| 100 | 1.691 |

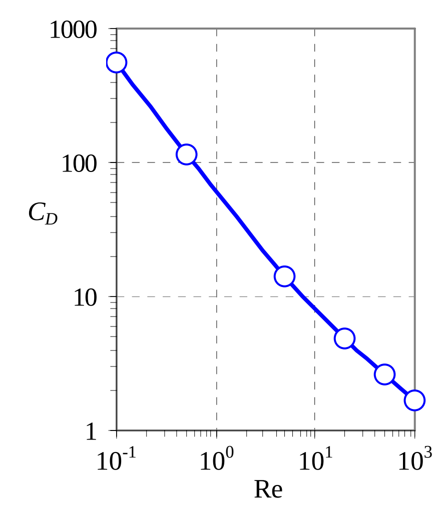



**FIGURE 1**
Drag coefficient plotted as a function of Reynolds number for creeping flow over a 2×1 ellipsoid.

***Analysis*** The velocity profile for creeping flow (Re < 1) shows a very slowly varying velocity from zero at the wall to $V$ eventually. At high Re, we expect a thin boundary layer and flow that accelerates around the body. However, in creeping flow, there is negligible inertia, and the flow does not accelerate around the body. Instead, the body has significant impact on the flow to distances very far from the body. As Re increases, the drag coefficient drops sharply, as expected based on experimental data (see Chap. 11). At the higher values of Re (here, for Re = 50 and 100), inertial effects are becoming more significant than viscous effects, and the velocity flow disturbance caused by the body is confined more locally around the body compared to the lower Reynolds number cases. If Re were to be increased even more, very thin boundary layers would develop along the walls. The data are also plotted in Fig. 1. The drop in drag coefficient with increasing Reynolds number is quite dramatic as Re ranges from 0.1 to 100. ($C_D$ decreases from more than 500 to nearly 1 in that range). Thus, we use a log-log scale in Fig. 1.

***Discussion*** Newer versions of FlowLab may give slightly different results.

## General CFD Problems

### 15-72

**Solution** We are to generate three different coarse grids for the same geometry and node distribution, and then compare the cell count and grid quality.

*Analysis* The three meshes are shown in Fig. 1. The node distributions along the edges of the computational domain are identical in all three cases, and no smoothing of the mesh is performed. The structured multi-block mesh is shown in Fig. 1a. We split the domain into four blocks for convenience, and to achieve cells with minimal skewing. There are 1060 cells. The unstructured triangular mesh is shown in Fig. 1b. There is only one block, and it contains 1996 cells. The unstructured quad mesh is shown in Fig. 1c. It has 833 cells in its one block. Comparing the three meshes, the triangular unstructured mesh has too many cells. The unstructured quad mesh has the least number of cells, but the clustering of cells occurs in undesirable locations, such as at the outlets on the right. The structured quad mesh seems to be the best choice for this geometry – it has only about 27% more cells than the unstructured quad mesh, but we have much more control on the clustering of the cells. Skewness is not a problem with any of the meshes.

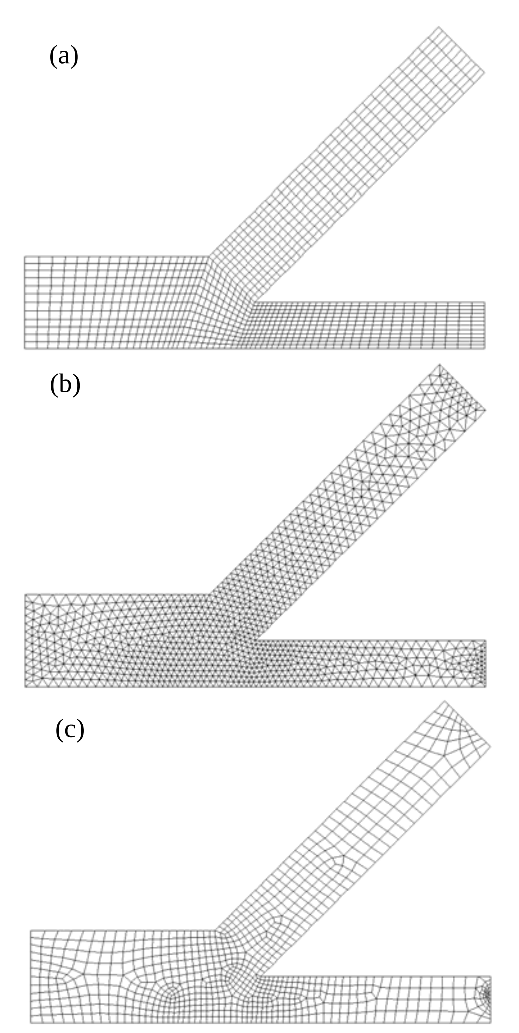



**FIGURE 1**
Comparison of three meshes: (a) structured multiblock, (b) unstructured triangular, and (c) unstructured quadrilateral.

*Discussion* Depending on the grid generation software and the specified node distribution, students will get a variety of results.

## 15-73

**Solution** We are to run a laminar CFD calculation of flow through a wye, calculate the pressure drop and how the flow splits between the two branches.

***Analysis*** We choose the structured grid for our CFD calculations. The back pressure at both outlets is set to zero gage pressure, and the average pressure at the inlet is calculated to be $-8.74 \times 10^{-5}$ Pa. The pressure drop through the wye is thus only $8.74 \times 10^{-5}$ Pa (a negligible pressure drop). The streamlines are shown in Fig. 1. For this case, 57.8% of the flow goes out the upper branch, and 42.2% goes out the lower branch.

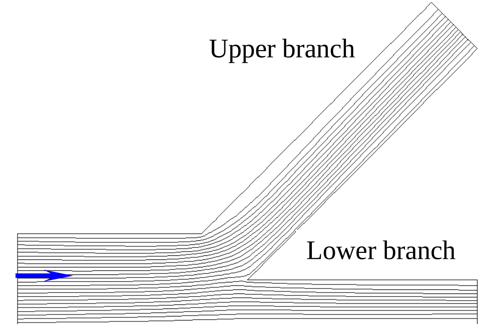



**FIGURE 1**
Streamlines for laminar flow through a wye.

***Discussion*** There appears to be some tendency for the flow to separate at the upper left corner of the branch, but there is no reverse flow at the outlet of either branch. This case is compared to a turbulent flow case in the following problem.

## 15-74

**Solution** We are to run a turbulent CFD calculation of flow through a wye, calculate the pressure drop and how the flow splits between the two branches.

***Analysis*** We choose the structured grid for our CFD calculations. The back pressure at both outlets is set to zero gage pressure, and the average pressure at the inlet is calculated to be -3.295 Pa. The pressure drop through the wye is thus 3.295 Pa (a significantly higher pressure drop than that of the laminar flow, although we note that the inlet velocity for the laminar flow case was 0.002 times that of the turbulent flow case). The streamlines are shown in Fig. 1. For this case, 54.4% of the flow goes out the upper branch, and 45.6% goes out the lower branch. Compared to the laminar case, a greater percentage of the flow goes out the lower branch for the turbulent case. The streamlines at first look similar, but a closer look reveals that the spacing between streamlines in the turbulent case is more uniform, indicating that the velocity distribution is also more uniform (more “full”), as is expected for turbulent flow.

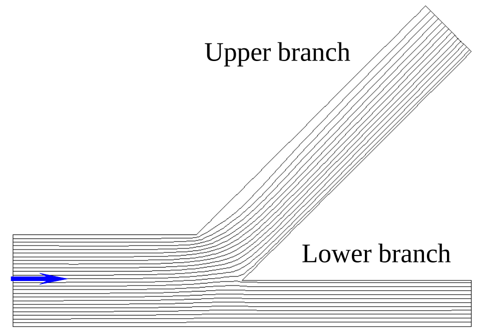



**FIGURE 1**
Streamlines for turbulent flow through a wye. The $k$-$\varepsilon$ turbulence model is used.

***Discussion*** There appears to be some tendency for the flow to separate at the upper left corner of the branch, but there is no reverse flow at the outlet of either branch.

## 15-75

**Solution** We are to keep refining a grid until it becomes grid independent for the case of a laminar boundary layer.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students’ results.

## 15-76

**Solution** We are to keep refining a grid until it becomes grid independent for the case of a turbulent boundary layer.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students’ results.

**15-77**

**Solution** We are to study ventilation in a simple 2-D room using CFD, and using a structured rectangular grid.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

**15-78**

**Solution** We are to repeat the previous problem except use an unstructured grid, and we are to compare results.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

**15-79**

**Solution** We are to use CFD to analyze the effect of moving the supply and/or return vents in a room.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

**15-80**

**Solution** We are to use CFD to analyze a simple 2-D room with air conditioning and heat transfer.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

**15-81**

**Solution** We are to compare the CFD predictions for 2-D and 3-D ventilation.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

**15-82**

**Solution** We are to use CFD to study compressible flow through a converging nozzle with inviscid walls. Specifically, we are to vary the pressure until we have choked flow conditions.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

**15-83**

**Solution** We are to repeat the previous problem, but allow friction at the wall, and also use a turbulence model. We are then to compare the results to those of the previous problem to see the effect of wall friction and turbulence on the flow.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

**15-84**

**Solution** We are to generate a low-drag, streamlined, 2-D body, and try to get the smallest drag in laminar flow.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

**15-85**

**Solution** We are to generate a low-drag, streamlined, axisymmetric body, and try to get the smallest drag in laminar flow. We are also to compare the axisymmetric case to the 2-D case of the previous problem.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

**15-86**

**Solution** We are to generate a low-drag, streamlined, axisymmetric body, and try to get the smallest drag in turbulent flow. We are also to compare the turbulent drag coefficient to the laminar drag coefficient of the previous problem.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

## 15-87

**Solution** We are to use CFD to study Mach waves in supersonic flow. We are also to compare the computed Mach angle with that predicted by theory.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

## 15-88

**Solution** We are to study the effect of Mach number on the Mach angle in supersonic flow, and we are to compare to theory.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

## Review Problems

## 15-89C

**Solution**

(*a*) ***False***: If the boundary conditions are not correct, if the computational domain is not large enough, etc., the solution can be erroneous and nonphysical no matter how fine the grid.
(*b*) ***True***: Each component of the Navier-Stokes equation is a transport equation.
(*c*) ***True***: The four-sided cells of a 2-D structured grid require less cells than do the triangular cells of a 2-D unstructured grid. (Note however, that some unstructured cells can be four-sided as well as three-sided.)
(*d*) ***True***: Turbulence models are *approximations* of the physics of a turbulent flow, and unfortunately are not universal in their application.

---

## 15-90C

**Solution** We are to discuss right-left symmetry as applied to a CFD simulation and to a potential flow solution.

***Analysis*** In the time-averaged CFD simulation, we are not concerned about top-bottom fluctuations or periodicity. Thus, top-bottom symmetry can be assumed. However, fluid flows do not have upstream-downstream symmetry in general, even if the geometry is perfectly symmetric fore and aft. In the problem at hand for example, the flow in the channel develops downstream. Also, the flow exiting the left channel enters the circular settling chamber like a jet, separating at the sharp corner. At the opposite end, fluid leaves the settling chamber and enters the duct more like an inlet flow, without significant flow separation. **We certainly cannot expect fore-aft symmetry in a flow such as this**.

On the other hand, potential flow of a symmetric geometry yields a symmetric flow, so it would be okay to cut our grid in half, invoking fore-aft symmetry.

***Discussion*** If unsteady or oscillatory effects were important, we should not even specify top-bottom symmetry in this kind of flow field.

---

**15-91C**

**Solution** We are to discuss improvements to the given computational domain.

***Analysis*** (*a*) Since Gerry is not interested in unsteady fluctuations (which may be unsymmetric), he could eliminate half of the domain. In other words, **he could assume that the axis is a plane of symmetry between the top and bottom of the channel**. Gerry's grid would be cut in size by a factor of two, leading to approximately half the required CPU time, but yielding virtually identical results.

(*b*) The fundamental flaw is that **the outflow boundary is not far enough downstream**. There will likely be flow separation at the corners of the sudden contraction. With a duct that is only about three duct heights long, it is possible that there will be reverse flow at the outlet. Even if there is no reverse flow, the duct is nowhere near long enough for the flow to achieve fully developed conditions. Gerry should extend the outlet duct by many duct heights to allow the flow to develop downstream and to avoid possible reverse flow problems.

***Discussion*** The inlet appears to be perhaps too short as well. If Gerry specifies a fully developed channel flow velocity profile at the inlet, his results may be okay, but again it is better to extend the duct many duct heights beyond what Gerry has included in his computational domain.

---

**15-92C**

**Solution** We are to discuss a feature of modern computer systems for which nearly equal size multiblock grids are desirable.

***Analysis*** The fastest computers are multi-processor computers. In other words, the computer system contains more than one CPU – a **parallel computer**. Modern parallel computers may combine 32, 64, 128, or more CPUs or *nodes*, all working together. In such a situation it is natural to let each node operate on one block. If all the nodes are identical (equal speed and equal RAM), the system is most efficient if the blocks are of similar size.

***Discussion*** In such a situation there must be communication between the nodes. At the interface between blocks, for example, information must pass during the CFD iteration process.

---

**15-93C**

**Solution** We are to discuss the difference between multigridding and multiblocking, and we are to discuss how they may be used to speed up a CFD calculation. Then we are to discuss whether multigridding and multiblocking can be applied together.

***Analysis*** *Multigridding* has to do with the resolution of an established grid during CFD calculations. With multigridding, **solutions of the equations of motion are obtained on a coarse grid first, followed by successively finer grids**. This speeds up convergence because the gross features of the flow are quickly established on the coarse grid (which takes less CPU time), and then the iteration process on the finer grid requires less time.

*Multiblocking* is something totally different. It refers to the **creation of two or more separate blocks or zones, each with its own grid**. The grids from all the blocks collectively create the overall grid. As discussed in the previous problem, multiblocking can have some speed advantages if using a parallel-processing computer. In addition, some CFD calculations would require too much RAM if the entire computational domain were one large block. In such cases, the grid can be split into multiple blocks, and the CFD code works on one block at a time. This requires less RAM, although information from the dormant blocks must be stored on disk or solid state memory chips, and then swapped into and out of the computer's RAM.

There is no reason why multigridding cannot be used on each block separately. Thus, **multigridding and multiblocking can be used together**.

***Discussion*** Although all the swapping in and out requires more CPU time and I/O time, for large grids multiblocking can sometimes mean the difference between being able to run and not being able to run at all.

---

## 15-94C

**Solution** We are to discuss why we should spend a lot of time developing a multiblock structured grid when we could just use an unstructured grid.

***Analysis*** There are several reasons why a structured grid is "better" than an unstructured grid, even for a case in which the CFD code can handle unstructured grids. First of all **the structured grid can be made to have better resolution *with fewer cells* than the unstructured grid**. This is important if computer memory and CPU time are of concern. Depending on the CFD code, the solution **may *converge more rapidly* with a structured grid**, and the results may be **more accurate**. In addition, by creating multiple blocks, we can more easily **cluster cells** in certain blocks and locations where high resolution is necessary, since we have much more control over the final grid with a structured grid.

***Discussion*** As mentioned in this chapter, time spent creating a good grid is usually time well spent.

---

## 15-95

**Solution** We are to calculate flow through a single-stage heat exchanger.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

## 15-96

**Solution** We are to study the effect of heating element angle of attack on heat transfer through a single-stage heat exchanger.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

## 15-97

**Solution** We are to calculate flow through a single-stage heat exchanger.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

## 15-98

**Solution** We are to study the effect of heating element angle of attack on heat transfer through a two-stage heat exchanger.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

## 15-99

**Solution** We are to study the effect of spin on a cylinder using CFD, and in particular, analyze the lift force.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

## 15-100

**Solution** We are to study the effect of spin speed on a spinning cylinder using CFD, and in particular, analyze the lift force as a function of rotational speed in nondimensional variables.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

## 15-101

**Solution** We are to study flow into a slot along a wall using CFD.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

## 15-102

**Solution** We are to calculate laminar flow into a 2-D slot, compare with irrotational flow theory, and with results of the previous problem, and discuss the vorticity field.

***Assumptions*** **1** The flow is steady and 2-D. **2** The flow is laminar.

***Analysis*** The flow field does not change much from the previous problem, except that a thin boundary layer shows up along the floor. The vorticity is confined to a region close to the floor – vorticity is negligibly small everywhere else, so the irrotational flow approximation is appropriate everywhere except close to the floor.

***Discussion*** The irrotational flow approximation is very useful for suction-type flows, as in air pollution control applications (hoods, etc.).

---

**15-103**

**Solution** We are to model the flow of air into a vacuum cleaner using CFD, and we are to compare the results to those obtained with the potential flow approximation.

***Analysis*** We must include a second length scale in the problem, namely the width $w$ of the vacuum nozzle. For the CFD calculations, we set $w$ = 2.0 mm and place the inlet plane of the vacuum nozzle at $b$ = 2.0 cm above the floor (Fig. 1). Only half of the flow is modeled since we can impose a symmetry boundary condition along the y-axis. We use the same volumetric suction flow rate as in the example problem, i.e., $\dot{V}/L = 0.314$ m$^2$/s, but in the CFD analysis we specify only *half* of this value since we are modeling half of the flow field.

Results of the CFD calculations are shown in Fig. 2. Fig. 2a shows a view of streamlines in the entire computational plane. Clearly, the streamlines far from the inlet of the nozzle appear as rays into the origin; from "far away" the flow feels the effect of the vacuum nozzle in the same way as it would feel a line sink. In Fig. 2b is shown a close-up view of these same streamlines. Qualitatively, the streamlines appear similar to those predicted by the irrotational flow approximation. In Fig 2c we plot contours of the magnitude of vorticity. Since irrotationality is defined by zero vorticity, these vorticity contours indicate where the irrotational flow approximation is valid – namely in regions where the magnitude of vorticity is negligibly small.

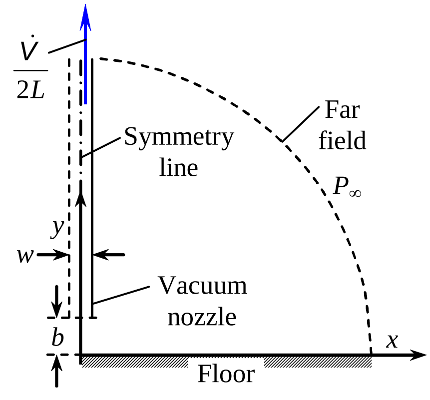
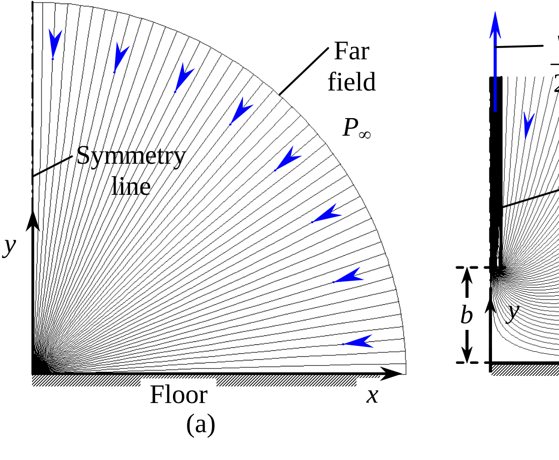
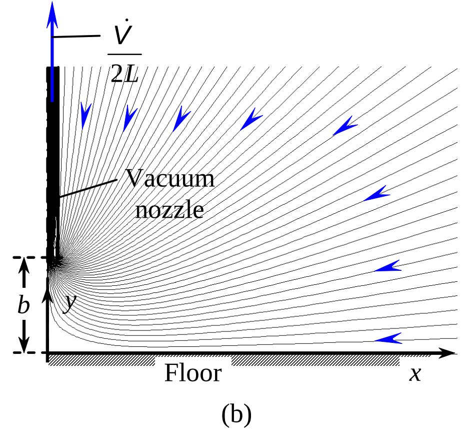



**FIGURE 1**
CFD model of air being sucked into a vertical vacuum nozzle; the $y$-axis is a line of symmetry (not to scale –the far field is actually much further away from the nozzle than is sketched here).



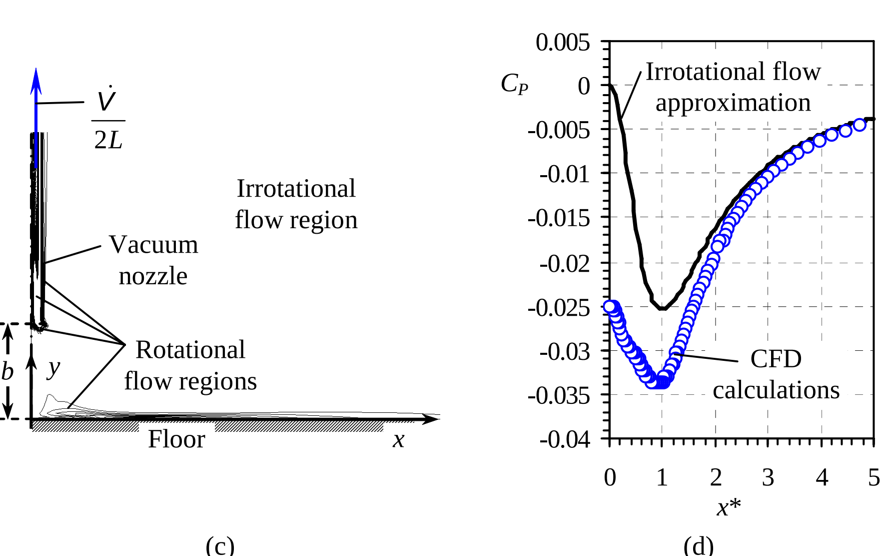




**FIGURE 2**
CFD calculations of flow into the nozzle of a vacuum cleaner; (a) streamlines in the entire flow domain, (b) close-up view of streamlines, (c) contours of constant magnitude of vorticity illustrating regions where the irrotational flow approximation is valid, and (d) comparison of pressure coefficient with that predicted by the irrotational flow approximation.

We see from Fig. 2c that vorticity is negligibly small everywhere in the flow field except close to the floor, along the vacuum nozzle wall, near the inlet of the nozzle, and inside the nozzle duct. In these

regions, net viscous forces are *not* small and fluid particles *rotate* as they move; the irrotational flow approximation is not valid in these regions. Nevertheless, it appears that the irrotational flow approximation is valid throughout the majority of the flow field. Finally, the pressure coefficient predicted by the irrotational flow approximation is compared to that calculated by CFD in Fig. 2d.

***Discussion*** For $x^*$ greater than about 2, the agreement is excellent. However, the irrotational flow approximation is not very reliable close to the nozzle inlet. Note that the irrotational flow prediction that the minimum pressure occurs at $x^* \approx 1$ is verified by CFD.

---

**15-104**

**Solution** We are to compare CFD calculations of flow into a vacuum cleaner for the case of laminar flow versus the inviscid flow approximation.

***Analysis*** Students will have varied results, depending on the grid generation code, CFD code, turbulence model, and their choice of computational domain, etc.

***Discussion*** Instructors can add more details to the problem statement, if desired, to ensure consistency among the students' results.

---

